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SMP - A SOLID MODELING PROGRAM
VERSION 2.0

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The attached Standard Bibliographic Page was inadvertently omitted from the report. Include the Standard Bibliographic Page as the last page in the report.

Issued November 1986

PREFACE

This document describes the features and utilities of the SOLID MODELING PROGRAM (SMP - Version 2.0).

This NASA Contractor Report supercedes an earlier SMP document (NASA Contractor Report 172473). The major enhancements to SMP Version 2.0 include a new "truss" primitive, a facility for computing mass properties and projected areas, an external program interface called the General Vehicle Synthesizer, Tektronix 4100 series terminal device interfaces, and a VAX/VMS computer system implementation of the software.

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Table of Contents

LIST OF TABLES.....	v
LIST OF FIGURES.....	vi
1 INTRODUCTION.....	1
2 MODELING PRIMITIVES.....	8
2.1 BASIC PRIMITIVES.....	11
2.1.1 BOX.....	15
2.1.2 CONE.....	16
2.1.3 SPHERE.....	18
2.1.4 PARABOLOID.....	20
2.1.5 TORUS.....	22
2.1.6 TRUSS.....	24
2.2 SWEPT PART.....	28
2.2.1 TRANSLATIONALLY SWEPT PARTS.....	30
2.2.2 ROTATIONALLY SWEPT PARTS.....	31
2.3 BOOLEAN PRIMITIVES.....	32
2.4 EXTERNALS.....	37
2.5 ASSEMBLIES.....	39
3 GEOMETRIC REPRESENTATION.....	42
3.1 PARTS FILE.....	45
3.1.1 BASIC PRIMITIVES PARTS-FILE.....	48
3.1.2 SWEPT PART PARTS-FILE DESCRIPTION.....	51
3.1.3 BOOLEAN PARTS-FILE DESCRIPTION.....	53
3.1.4 EXTERNAL PARTS-FILE DESCRIPTION.....	54
3.1.5 ASSEMBLY PARTS-FILE DESCRIPTION.....	56
3.2 GEOMETRY FILE.....	57
4 COMMANDS.....	62
4.1 READ COMMAND LEVEL.....	75
4.2 WRITE COMMAND LEVEL.....	77
4.3 EDIT COMMAND LEVEL.....	80
4.3.1 ADD.....	82
4.3.2 MODIFY.....	84
4.3.3 DELETE.....	90
4.3.4 RESTORE.....	93
4.3.5 PRINT.....	96
4.3.6 COPY.....	100
4.4 DISPLAY COMMAND LEVEL.....	102
4.4.1 RESET COMMAND.....	104
4.4.2 MENU COMMAND.....	105
4.4.3 DRAW COMMAND.....	109
4.4.4 FOUR VIEW COMMAND.....	110
4.4.5 ZOOM COMMAND.....	112

TABLE OF CONTENTS (cont.)

4. 4. 6	HIDDEN SURFACE COMMAND.....	114
4. 4. 7	COMPOSITE HIDDEN SURFACE.....	121
4. 4. 8	GRAPHICS EDITOR.....	122
4. 4. 8. 1	PICK A PART.....	130
4. 4. 8. 2	PICK A PART AND EDIT.....	130
4. 4. 8. 3	PRINT VERTEX COORDINATES.....	130
4. 4. 8. 4	TRANSLATE A PART.....	130
4. 4. 8. 5	ROTATE A PART.....	131
4. 5	MODEL ANALYSIS COMMAND LEVEL.....	136
4. 5. 1	MASS PROPERTIES.....	137
4. 5. 2	PROJECTED AREAS.....	143
5	MISCELLANEOUS COMMANDS.....	146
5. 1	GENERAL VEHICLE SYNTHESIZER.....	147
5. 1. 1	MODIFY THE DENSITY OF ASSEMBLIES.....	152
5. 1. 2	GVS MASS PROPERTIES.....	153
5. 1. 3	GVS AREAS AND BLOCKAGE.....	155
5. 1. 4	LOCATE ADDED NODES.....	158
5. 1. 4. 1	ADD A NODE.....	159
5. 1. 4. 2	DELETE A NODE.....	162
5. 1. 4. 3	MODIFY A NODE.....	163
5. 1. 4. 4	DISPLAY ADDED NODES.....	164
5. 1. 5	GVS INITIALIZATION.....	165
5. 1. 6	WRITE GVS OUTPUT FILE.....	166
5. 1. 7	READ GVS COMMON DATA FILE.....	168
5. 1. 8	WRITE GVS BATCH COMMAND FILE.....	173
5. 2	DIMENSIONS BETWEEN PARTS.....	176
6	SOFTWARE AND USER ENVIRONMENTS.....	177
6. 1	SOFTWARE ENVIRONMENT.....	178
6. 1. 1	SOURCE-LEVEL LANGUAGE.....	179
6. 1. 2	HOST INTERFACES.....	180
6. 1. 2. 1	PRIME/PRIMOS HOST.....	181
6. 1. 2. 2	VAX/VMS HOST DEPENDENCIES.....	183
6. 1. 3	DEVICE INTERFACES.....	184
6. 1. 3. 1	ALPHANUMERIC TERMINAL.....	186
6. 1. 3. 2	ADAGE 3000 COLOR RASTER.....	187
6. 1. 3. 3	ADVANCED ELECTRONICS DESIGN.....	188
6. 1. 3. 4	INTELLIGENT SYSTEMS.....	189
6. 1. 3. 5	TEKTRONIX 4100 SERIES.....	190
6. 1. 3. 6	TEKTRONIX 4010 SERIES.....	192
6. 2	USER ENVIRONMENT.....	193
6. 2. 1	VAX/VMS ENVIRONMENT.....	194
6. 2. 2	PRIME/PRIMOS ENVIRONMENT.....	197
	APPENDIX A: SMP IMPLEMENTATION.....	198
	APPENDIX B: TRUSS FORMATION.....	204
	REFERENCES.....	208

LIST OF TABLES

TABLE 1	BASIC PRIMITIVE ATTRIBUTES	49
TABLE 1A	TRUSS ATTRIBUTES	50
TABLE 2	BASIC SWEPT PART ATTRIBUTES	52
TABLE 3	GEOMETRY FILE DESCRIPTION	60
TABLE 4	SMP COMMAND HIERARCHY	64
TABLE 5	MASS PROPERTY FUNCTIONS	138
TABLE 6	COMMON FILE DESCRIPTION	172
TABLE 7	SMP DEVICE SUPPORT	185
TABLE 8	SMP SOFTWARE MODULES	199
TABLE 9	SMP UTILITY AND DEVICE LIBRARIES	200

LIST OF FIGURES

FIGURE 1	SINGLE KEEL SPACE STATION	4
FIGURE 2	ROBOT ARM	5
FIGURE 3	GEOMETRIC MODELING PRIMITIVES	12
FIGURE 4	TRUSS	13
FIGURE 5	BOOLEAN OPERATION	34
a	HIDDEN LINE	35
b	SHADED IMAGE	47
FIGURE 6	PARTS-FILE	61
FIGURE 7	GEOMETRY-FILE	98-99
FIGURE 8	EDITOR PRINT FORMAT	
FIGURE 9	DISPLAY OPTIONS	108
a	NO BACK FACE CULL	108
b	BACK FACE CULL	108
c	DASH BACK FACES	108
d	SHRINK FACES	111
FIGURE 10	SPACE STATION FOUR VIEW	
FIGURE 11	SURFACE SHADED IMAGE	120
a	FLAT SHADING	120
b	SMOOTH SHADING	
FIGURE 12	GRAPHICAL PART SELECTION	126
a	XY PLANE VIEW	127
b	YZ PLANE VIEW	
FIGURE 13	GRAPHICAL PART ROTATION	133
a	UNROTATED FOUR VIEW	134
b	REFERENCE VERTEX AND LINE SELECTION	135
c	ROTATED FOUR VIEW	
FIGURE 14	MASS PROPERTIES OUTPUT	142
FIGURE 15	PROJECTED AREAS OUTPUT	145

LIST OF FIGURES (cont.)

FIGURE 16	PROJECTED AREAS AND BLOCKAGE FACTOR OUTPUT	157
FIGURE 17	BATCH PROCESSING	175
FIGURE 18	LINKING SMP	202-203

1 INTRODUCTION

The Solid Modeling Program (SMP) provides the capability to model complex solid objects through the composition of "primitive" geometric entities. In addition to the construction of solid models, SMP has extensive facilities for model editing, display and analysis. The geometric model produced by the software system can be output in a format compatible with existing analysis programs such as PATRAN-G.

SMP originated as a graphics postprocessor for an advanced spacecraft concepts preliminary design program [1]. Since those early beginnings, SMP has been utilized in a stand alone mode to model proposed space station configurations and large antenna designs [2]. A model of a single keel space station is shown in FIGURE 1. Because the system models objects through the manipulation of basic shapes, it can be employed to support a variety of applications.

A solid model generated by SMP is comprised of a collection of geometric "primitives" (parts). The system provides the designer with a basic set of primitive parts and the capability for defining new primitives. The present version of the SMP software supports six primitives: "boxes", "cones", "spheres", "paraboloids", "tori" and "trusses". The

user defines a primitive part by specifying the dimension and construction attributes required for a given part type. By varying the construction attributes for certain primitives, numerous additional shapes can be represented. For example, a "cylinder" is a special case of a "cone" where both radii are equal. Regularly shaped geometric objects can also be generated through the application of translational or rotational "sweeping". In this case, a two dimensional profile of node points is user supplied, and the modeller "sweeps" out the three dimensional solid object. New primitives can also be generated by operating on existing pairs of primitives with Boolean ("set") operations of intersection, union, and difference. Finally, primitives can be created external to the SMP software. The only restriction for "external" parts is that the SMP geometry format be rigidly followed. In certain instances, it may be advantageous to group related or frequently used primitives, and later reference this group as a single entity. The mechanism for grouping parts in SMP is the "assembly". A representative geometric model illustrating most major part classifications is shown in FIGURE 2. The designer can apply a transformation to any category of part to insure proper orientation. The details for creating each of the major primitive types is presented later in the document.

SMP provides a facility for approximating the mass properties of the resulting solid model. Because of SMP's association with spacecraft design, the mass property calculations have been expanded to include projected area approximations. The analysis capabilities of SMP, including interfaces to existing analysis programs, are discussed in later sections.

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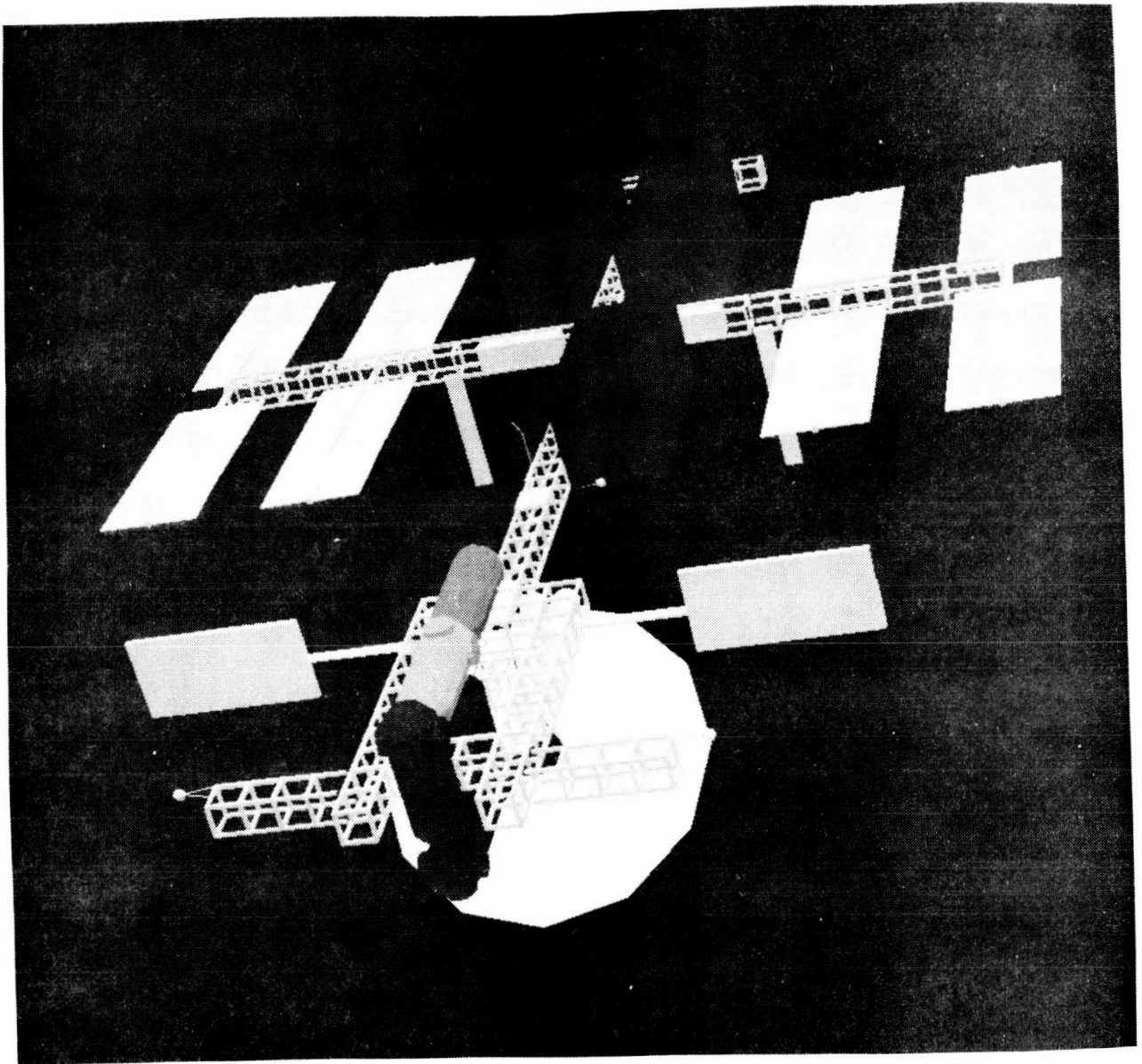


FIGURE 1
SINGLE KEEL SPACE STATION

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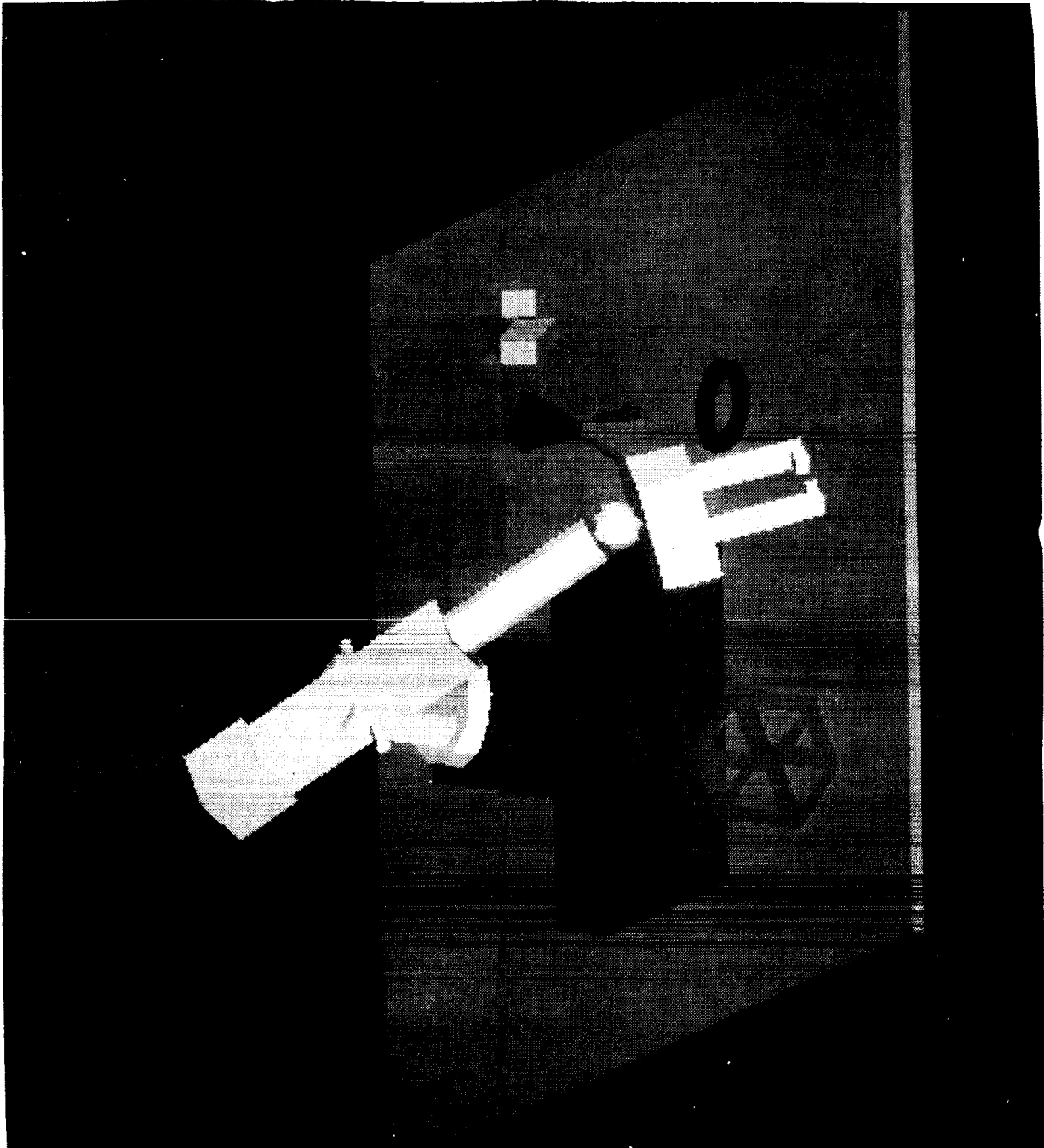


FIGURE 2
ROBOT ARM

One of the major concerns in any solid modeling package is the selection of an appropriate geometric representation scheme. The contemporary schemes for representing solid data are outlined in [3]. SMP employs a dual representation scheme. Internal to the software system, the solid objects are defined by a hybrid boundary representation whereby each part is described by a collection of three dimensional vertices and the associated "connectivity" between the vertices. The boundary is, therefore, comprised of a set of planar facets ("faces"). Unless explicitly requested, the designer need not be concerned with vertices and connectivity. For this reason, a hybrid constructive solid geometry (CSG) scheme is invoked for representing the solid model externally. SMP does not utilize the CSG tree but instead, generates a parts list where each list entry contains the minimal information necessary for reconstructing and orienting the primitive. Both the schemes for geometric representation, together with their respective input and output interfaces, will be detailed in a later section of this report.

The SMP software is structured as a hierarchy with each level being associated with a set of program commands. The system is menu and command driven with an online help facility available at each level. The highest level menu addresses the areas of: reading and writing the solid model geometry,

primitive editing, model display, and model analysis. If a specific command has options, the user will be prompted as required. Errors will be detected and corrected whenever possible. In some cases, the corrections will require some additional action on the part of the user. In either case, a terminal message will inform the user as to the nature of the error and instruct the user as to an appropriate action. The command levels, sublevels, individual commands, and command options will be described and demonstrated in a subsequent section.

SMP provides a natural approach to creating, manipulating, and displaying solid objects. Because the user unambiguously defines a primitive with only dimension (length, radius, ..., etc.) and orientation (rotate, scale, and/or translate) information, user input is kept to a minimum. Through the utilization of externally created primitives, the designer can model objects whose components are irregularly shaped. The intent of this report is to describe the features of the modeller software and to demonstrate their utility.

2 MODELING PRIMITIVES

SMP models solid objects through the construction and orientation of three dimensional (3-D) "primitives". The present version of the modeller software supports five categories of primitives:

- (1) basic primitives
- (2) swept primitives
- (3) Boolean primitives
- (4) external primitives
- (5) assemblies

The basic primitives include the "box", "cone", "sphere", "parabaloid", "torus", and "truss"; swept primitives include 3-D objects generated by either the translational or rotational "sweeping" of a two dimensional (2-D) profile curve; Boolean primitives are the result of the application of one of the "set" operations of union, intersection, or difference to an existing primitive pair; external primitives are 3-D geometric entities created external to SMP; and assemblies provide a mechanism for grouping existing primitives. The intent of this section is to detail the specification and construction of each primitive type from the viewpoint of the user and the internals of the modeling package.

Once a primitive has been created, it must be positioned and oriented in relation to the entire model. SMP supports this

requirement by permitting a local transformation to be specified for every primitive. The local transformation takes the form of a rotation, scaling, and translation in each of the x, y, and z directions. SMP employs a standard "right-handed" coordinate system. This nine parameter transformation becomes an attribute of the primitive definition and is carried along with the part throughout its existence. Because the local transformation is retained, the individual transformations are order dependent. The predetermined order is rotation, scaling, and translation on the x, y, and z axes, respectively. This order puts some additional burden on the user, particularly with respect to selecting the proper rotation angles; but a facility for the arbitrary input of rotation angles has been included (see section 4.3.2). If the user chooses not to apply a local transformation, a default identity transformation is supplied. A notable exception to this rule is the initialization of the translation values. For certain primitive types the part may be automatically translated to a position more suitable for local orientation. More information concerning initial positioning will be given in the subsections pertaining to the individual primitives.

Although the specification requirements vary between primitives, two pieces of information are always required.

The user can specify an 80 character description for each primitive part (the default is all "blanks"). The user must specify a color value from 1 to 7 for every primitive. The number to color correspondence is as follows:

- 1 - red
- 2 - green
- 3 - yellow
- 4 - blue
- 5 - magenta
- 6 - cyan
- 7 - white

The color values are ignored on a monochromatic device.

The following subsections describe the specification for every primitive. The reader should note that all primitive attributes can be modified via the "editing" facilities resident within SMP (see section 4).

2.1 BASIC PRIMITIVES

Solid models are very often comprised of basic 3-D geometric shapes. In order to take advantage of this regularity, six basic primitives have been incorporated into the modeling software: "box", "cone", "sphere", "paraboloid", "torus", and "truss". These primitives are completely defined through the specification of the appropriate dimension and construction parameters. Although the number of basic primitives is relatively small, altering the construction parameters expands the number of basic shapes. For example, an entire class of conical shapes can be generated by varying the "number of sides" (planar sections). FIGURE 3 and FIGURE 4 exemplify the utility of the six "basic" primitives.

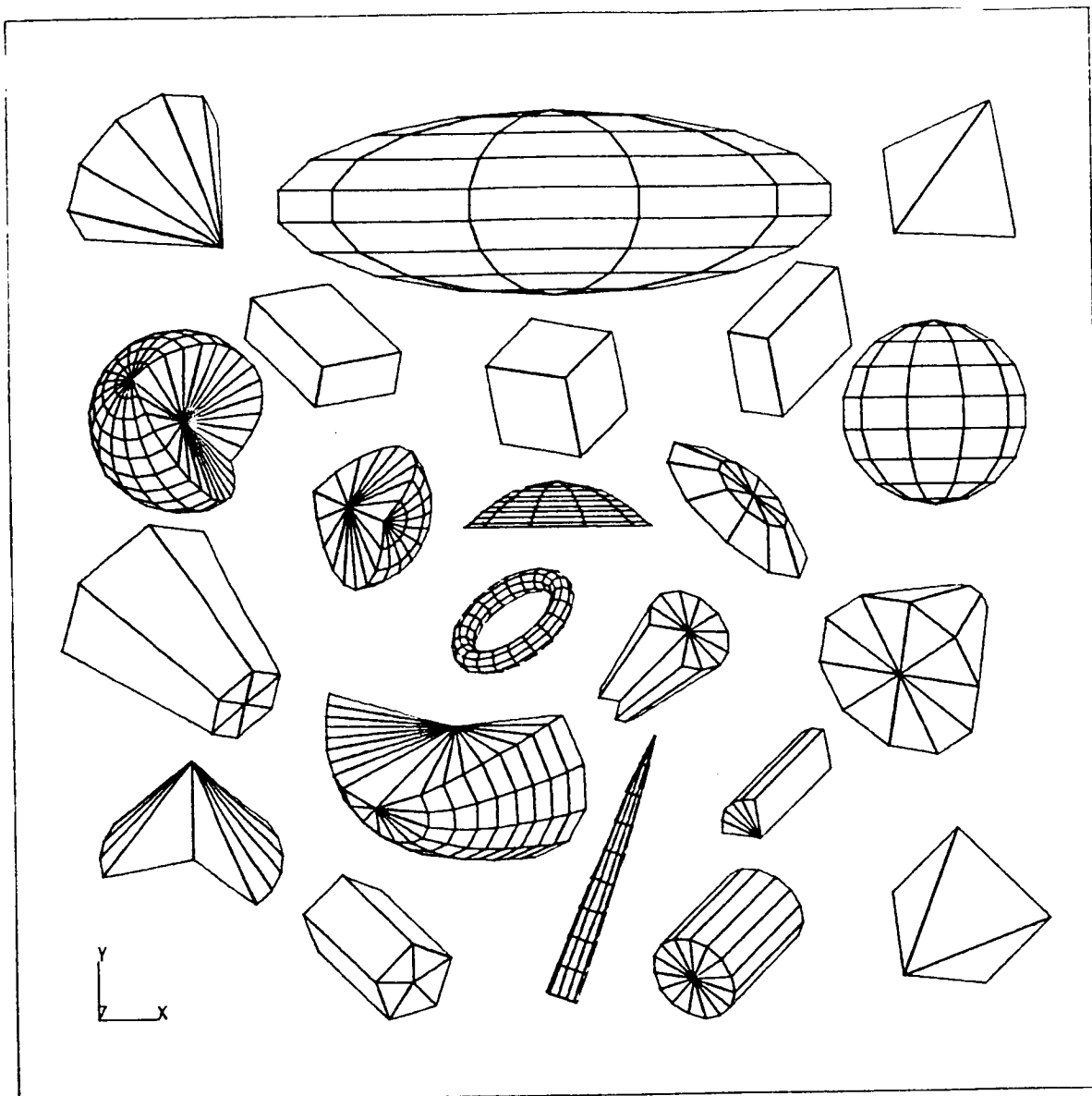


FIGURE 3
GEOMETRIC MODELING PRIMITIVES

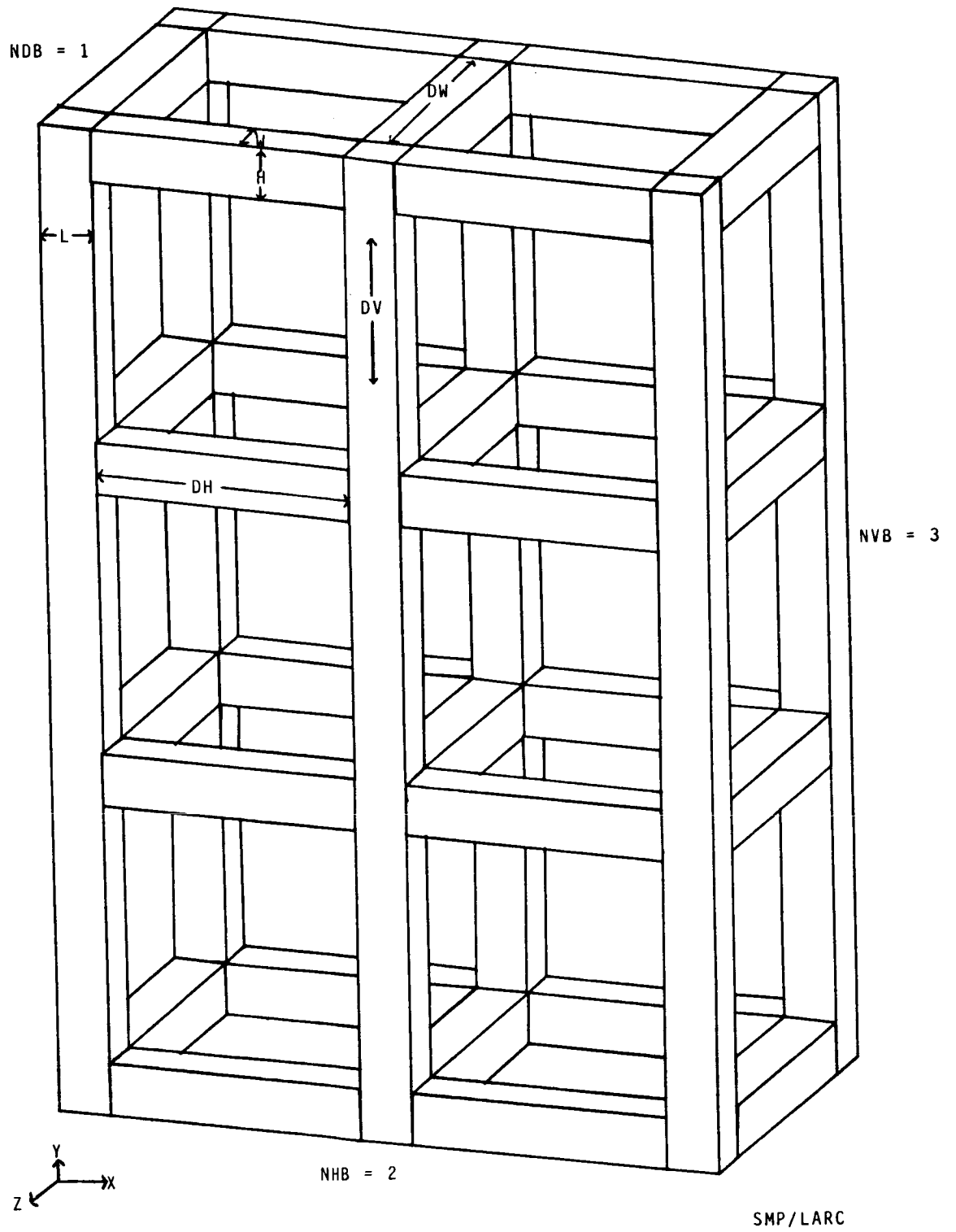


FIGURE 4
TRUSS

Once the designer has selected the desired basic primitive, the SMP software will prompt him/her for the required dimension and construction attributes. Internal error checking is provided to prohibit the user from entering illegal primitive specifications. The following subsections define the specification data required for each basic primitive.

2.1.1 BOX

The "box" is a rectangular hexahedron. The required dimensions are length, width, and height (real values > 0). Creation of the boundary representation for the "box" is accomplished through the determination of eight vertices such that the specified length is along the x-axis, width is along the z-axis, height is along the y-axis; and the "centroid" of the "box" is at the origin (0,0,0). In addition to the normal uses of a "box", it is useful for simulating other shapes when detail is not significant because of the minimal amount of information required for its boundary representation (see section 3.2).

2.1.2 CONE

The "cone" is a general form used to describe any type of cone, truncated cone, or cylinder. The dimensions of a "cone" are completely defined by supplying two radii of the two circles representing its ends (real values ≥ 0 , where both cannot equal 0.) and the length (real value > 0). However, the boundary representation for the "cone" is composed of planar subsections.

The "cone" is constructed by revolving (rotationally sweeping) a line segment in a circular path around the x-axis. The endpoints of the line segment are defined by the right most ("positive") and left most ("negative") radii of the circular ends. The slope of the line segment is determined by the relative difference of the two radii. Performing the revolution in discrete increments generates the planar subsections. If an end radius is greater than 0, the end is "capped" by forming triangular subsections emanating from the center of the circle. The resulting "cone" has its "centroid" at the origin with its length parallel to the x-axis.

The user controls the number of increments (and thus the number of planar subsections) by supplying the number of sides for the "cone" (integer value > 2). The boundary

representation limitations for a single primitive (see section 3.2) restrict the user's choice to:

$$(\text{number of sides} \times 2) + 2 \leq 300$$

Increasing the number of sides tends to "smooth" the "cone's" surfaces but also increases the amount of data in its boundary representation. A reasonable number for the number of sides appears to be 12.

Wedges of a "cone" can be created by varying the revolution angle of the sweep (real value $0. < \theta \leq 360$). An angle of 360 will result in the generation of a complete revolution. Angles less than 360 will result in some relative portion of the "cone" being generated. The interior of the partial "cones" will be constructed by joining the midpoints of the "endcaps".

The "cone" is perhaps the most versatile primitive part. By specifying one radius equal to zero, a true cone is generated. Unequal radii result in a truncated cone and equal radii produce a cylinder. By controlling the radii and the number of sides, special geometric structures such as tetrahedra and pyramids may be constructed. FIGURE 3 illustrates several applications of the "cone" primitive.

2.1.3 SPHERE

The dimensions of a "sphere" are completely defined by supplying the radius (real value >0). However, the boundary representation for the "sphere" is composed of planar subsections ("faces"). The "sphere" is constructed by revolving (rotationally sweeping) the circle, of the given radius and centered at the origin, around the y-axis. The planar subsections are generated by performing the revolution in discrete increments. The user controls the number of increments (and thus the number of planar subsections) by supplying the number of latitude and longitude lines (integers > 2) sectioning the "sphere". The boundary representation limitations for a single primitive (see section 3.2) restrict the user's choice to:

$$(\text{number latitudes} - 2) \times (\text{number longitudes}) + 2 \leq 300$$

Increasing the number of latitudes and/or longitudes tends to "smooth" the "sphere's" surfaces but also increases the amount of data in its boundary representation. A reasonable number for both parameters appears to be 10.

Wedges of a "sphere" can be created by varying the revolution angle of the sweep (real value $0. < \theta \leq 360$). An angle of 360 will result in the generation of a complete "sphere". Angles less than 360 will result in some relative portion of

the "sphere" being generated. The interior of the partial "spheres" will be constructed from triangular subsections emanating from the "sphere's" center.

The "sphere" is initially positioned at its center which is the origin.

Ellipsoids can be created by supplying nonequivalent scale factors in the local transformation.

2.1.4 PARABOLOID

The dimensions of a "paraboloid" are completely defined by providing the parameters for the equation of a parabola $y^2=4px$ where x is the length of the parabola and p is the distance from the vertex to the focus. However, the boundary representation for the "paraboloid" is composed of planar subsections ("faces"). Analogous to the "cone" and "sphere", the paraboloid is constructed by revolving (rotationally sweeping) the specified parabola around the x-axis. The planar subsections are generated by performing the revolution in discrete increments.

The user controls the number of increments (and thus the number of planar subsections) by supplying the number of latitude and longitude lines (integer values > 2) sectioning the "paraboloid." The boundary limitations for a single primitive (see section 3.2) restricts the users choice to:

$$(\text{number latitudes} - 1) \times (\text{number longitudes}) + 1 \leq 300$$

Increasing the number of latitudes and/or longitudes tends to "smooth" the "paraboloid's" surfaces but also increases the amount of data in its boundary representation. A reasonable number for both parameters appears to be 10.

Wedges of a "paraboloid" can be created by varying the

revolution angle of the sweep (real value $0. < \theta \leq 360.$). An angle of 360 will result in the generation of a complete "paraboloid." Angles less than 360 will result in some relative portion of the "paraboloid" being generated. The interior of the partial "paraboloids" will be constructed from triangular subsections emanating from the "paraboloid's" center.

Elliptical "paraboloids" may be generated by supplying nonequivalent y and z scale factors.

2.1.5 TORUS

The dimensions of a "torus" are completely described by supplying the inner radius and the outer radius. However, the boundary representation for the torus is composed of planar subsection ("faces"). For construction of the "torus", a circle (in the xy-plane) with diameter

$$D = \text{outer radius} - \text{inner radius}$$

and center

$$(-(\text{inner radius} + D/2), 0, 0)$$

is determined. This circle is then revolved (rotationally swept) around the y-axis to form the torus. The planar subsections are generated by the revolution in discrete increments. The resulting torus has its centroid at the origin and its "hole" perpendicular to the y-axis.

The user controls the number of increments (and thus the number of planar subsections) by supplying a number of "sections" (for the revolution) and a number of "sides" (for the circle). The boundary representation limitations for a single primitive (see section 3.2) restrict the user's choice to:

$$(\text{number of sides}) \times (\text{number of sections}) \leq 300$$

Increasing the number of sides and/or sections tends to "smooth" the "torus" surfaces but also increases the amount

of data in its boundary representation. A reasonable number for both parameters appears to be 12.

The start angle determines the location of the first increment for the initial circle. This parameter has little effect on "tori" with many sides but does affect the appearance of "tori" with number of sides less than 6. The usual value for the start angle is 0.

2.1.6 TRUSS

The "truss" is a collection of elements or supports grouped together to form a 3-D lattice structure. The dimensions of a "truss" are defined by the dimensions of the support members, the dimensions of the space between the support members (bays), and the number of bays in the horizontal, vertical, and depth directions.

The boundary representation for the "truss" is composed of planar subsections or faces, and is created by forming the boundary representation for each support member and accounting for the dimensions of each bay and the number of bays in each direction. Each support member is represented in a fashion similar to the "box". In order to understand the boundary representation of a "truss", it is necessary to first understand the boundary representation of the "box" (see section 2.1.1), and then how the parameters used to define the truss contribute to the boundary representation definition.

Nine parameters are required to define the "truss". Three describe the dimensions of the support members: the length of a truss element (L), the height of a truss element (H), and the width of a truss element (W) (all real values > 0). The distances between truss elements define the size of the bays

as well as the truss. These three parameters are the horizontal distance measured along the x-axis between truss elements (DH), the vertical distance measured along the y-axis between truss elements (DV), and the depth distance measured along the z-axis between truss elements (DW) (all real values > 0). The final three parameters defining the truss are the number of horizontal bays (NHB) counted along the x-axis, the number of vertical bays (NVB) counted along the y-axis, and the number of depth bays (NDB) counted along the z-axis (all integer values > 0). FIGURE 4 illustrates the nine "truss" parameters on a truss with two horizontal bays, three vertical bays, and one depth bay. The boundary representation for each support member is derived from subsets of these nine parameters. A "truss" may be specified as closed or open to enable the user to join "trusses" together. In order to maintain uniform dimensions when joining two "trusses", one "truss" is required to be open. The user is given six options as to the location of the open end.

The truss elements or supports may be thought of as either vertical, horizontal, or depth supports. Generally, vertical supports have their maximum length in the y-direction, horizontal supports in the x-direction, and depth supports in z. The boundary representation for each type of support varies slightly, but is basically analagous to that of a

"box". The lower left corner of the "truss" is positioned at the origin to facilitate the calculation of translation values. In this respect, the "truss" primitive is different from other primitives where the origin defines the "centroid" of the part. The most noticeable effect of this difference occurs when a rotation is performed; the "truss" rotates about its lower left corner instead of about its "centroid".

Horizontal supports are constructed parallel to the x-axis, vertical supports parallel to the y-axis, and depth supports parallel to the z-axis. Each support is constructed by incrementing x and y in the positive direction and decrementing z in the negative direction.

The boundary representation limitations for a single primitive (see section 3.2) restrict the number of bays as follows:

$$6 * (NHB + 1) * (NDB + 1) + 4 * (NDB + 1) * (NVB + 1) * NHB + 4 * (NHB + 1) * (NVB + 1) * NDB \leq 600$$

and,

$$8 * (NHB + 1) * (NDB + 1) + 8 * NHB * (NDB + 1) * NVB + 8 * NVB * NDB \leq 300.$$

The first expression limits the number of faces, and the second limits the number of points. If the number of bays specified exceeds these limits, the following action is taken: The number of depth bays is reduced to 1, and the user is

informed with an error message. The number of points and faces is recomputed and, if the limits are again exceeded, the number of horizontal bays is reduced to 1. The same procedure is followed, and if the limits are exceeded a third time, the number of vertical bays is reduced to 1. A detailed discussion on how the boundary representation for a truss is formed can be found in Appendix B. This discussion may be omitted by the general user.

2.2 SWEPT PART

A swept part is the 3-D object (solid, shell, or surface) defined by moving ("sweeping") a 2-D profile along a linear or circular path. SMP supports two types of swept parts: translational (linear path) and rotational (circular path); twisted translational and spiral swept parts are not included.

The input for swept parts deviates significantly from the attribute values used in the "basic" primitives (see section 2.1) since the user must explicitly specify the (X,Y) coordinates that define the 2-D profile curve. The user will be notified as to the maximum number of points allowed; this number may be dependent on previously defined part attributes. A minimum of two points is required to generate a swept part. Curves are defined in the plane $Z=0$ and must be defined in a clockwise order to insure consistency in the boundary representation (see section 3.2). To define a "closed" curve or a loop, the last (X,Y) point must be coincident with the first (X,Y) point entered. Care should be taken when creating an "open" swept curve since the resultant geometric shape is essentially "hollow" (i.e. not solid).

The input for the swept parts allows the user to "cap" the open ends of the part. A flag is used to signal the selection

of "end capping": 1 for "end caps" or a 2 for no "end caps". There is no general triangularization algorithm to implicitly construct the "end caps". Consequently, if "end caps" are selected, the user is requested to enter the (X,Y) coordinates of a center point. This feature should not be used if there does not exist a point within the interior of the closed curve such that straight lines can be drawn from each (input) point on the curve to the interior point without any of the lines intersecting the curve itself. The straight lines define the edges of triangular subsections which together form the "end cap".

The "centroid" for a swept part is the center of the smallest bounding rectangular hexahedron (with sides parallel to the coordinate axis) containing the part. The "centroid" value replaces the translation values in the transformation matrix when the part is initially created.

2.2.1 TRANSLATIONALLY SWEPT PARTS

The path of the translationally swept part is defined by entering the (X,Y,Z) coordinates of the new origin. The effect of this new origin (X,Y,Z) is that each coordinate $(x,y,0)$ on the profile curve defined by the user will be "connected" along a linear path to a new point $(x+X, y+Y, Z)$. One restriction placed on the new origin is that the Z value may not equal zero since the resulting part would have no "thickness". Currently, the maximum number of points for the user-defined curve is 30.

2.2.2 ROTATIONALLY SWEPT PARTS

Input for the rotationally swept part includes the number of sides (integer value ≥ 1) and the revolution angle ($0 < \theta \leq 360$). The profile curve is rotated counter-clockwise around the positive X axis in increments corresponding to the number of sides. For best results the curve should have positive Y values and only the end points of an "open" curve or at most one point on a "closed" curve should have Y values equal to zero. "End capping" is only recognized on wedges (i.e. revolution angle less than 360). The maximum number of points for the user-defined curve is inversely proportional to the number of sides, so if the number of side is decreased then more points may be entered. The calculation of the maximum number of points is as follows:

Maximum Number of Points = Minimum(30,
300/(Number Sides + 1),
600/(Number Sides + 2))

2.3 BOOLEAN PRIMITIVES

Boolean primitives are created by "joining" an existing pair of primitives through one of the "set" (or Euler) operations of: union, intersection, or difference. Several prototype and commercially available solid modeling packages use the Boolean operators as the fundamental mechanism for generating objects [3-4]. The algorithm used for combining primitives with one of the "set" operations is dependent on the internal representation of the geometric data. Because SMP uses a hybrid boundary representation scheme (see section 3.2), it is the boundary representations for each primitive that are computationally "intersected". The algorithm used within the modeller is based on the work of Maruyama [5], Parent [6], and Carlson [7].

The input required for creating a Boolean primitive consists solely of specifying the operator and the operands. The operator may be any one of the three basic "set" operations of:

intersection
union
difference

The operands are specified one at a time and must be selected from among the "active" (see section 4.3.5) primitives which currently comprise the user's model. All primitives with the

exception of assemblies (see section 2.5) are legal operands. Existing Boolean primitives can be used as operands for the purpose of creating new primitives.

Reversing the order of the operands usually results in the creation of a different primitive. For the cases of union and intersection, reversing the operands generates primitives of similar shapes but different boundary representations. However, reversing the operand order for the difference operator creates uniquely different shaped objects.

An example of the four basic set operations is shown in FIGURE 5 in both hidden line 5(a) and shaded image 5(b) renderings. The "basic" primitives (see section 2.1) of the "box" and "cone" are used to show the penetration of one primitive by another. The following operations are shown:

box (intersect)	cone	(upper left)
box (difference)	cone	(upper right)
box (union)	cone	(lower left)
cone (difference)	box	(lower right)

The extra construction lines on the "faces" of the "box" are due to the restrictions of the data representation scheme (see section 3.2). Specifically, a "face" can be comprised of no more than 4 "vertices".

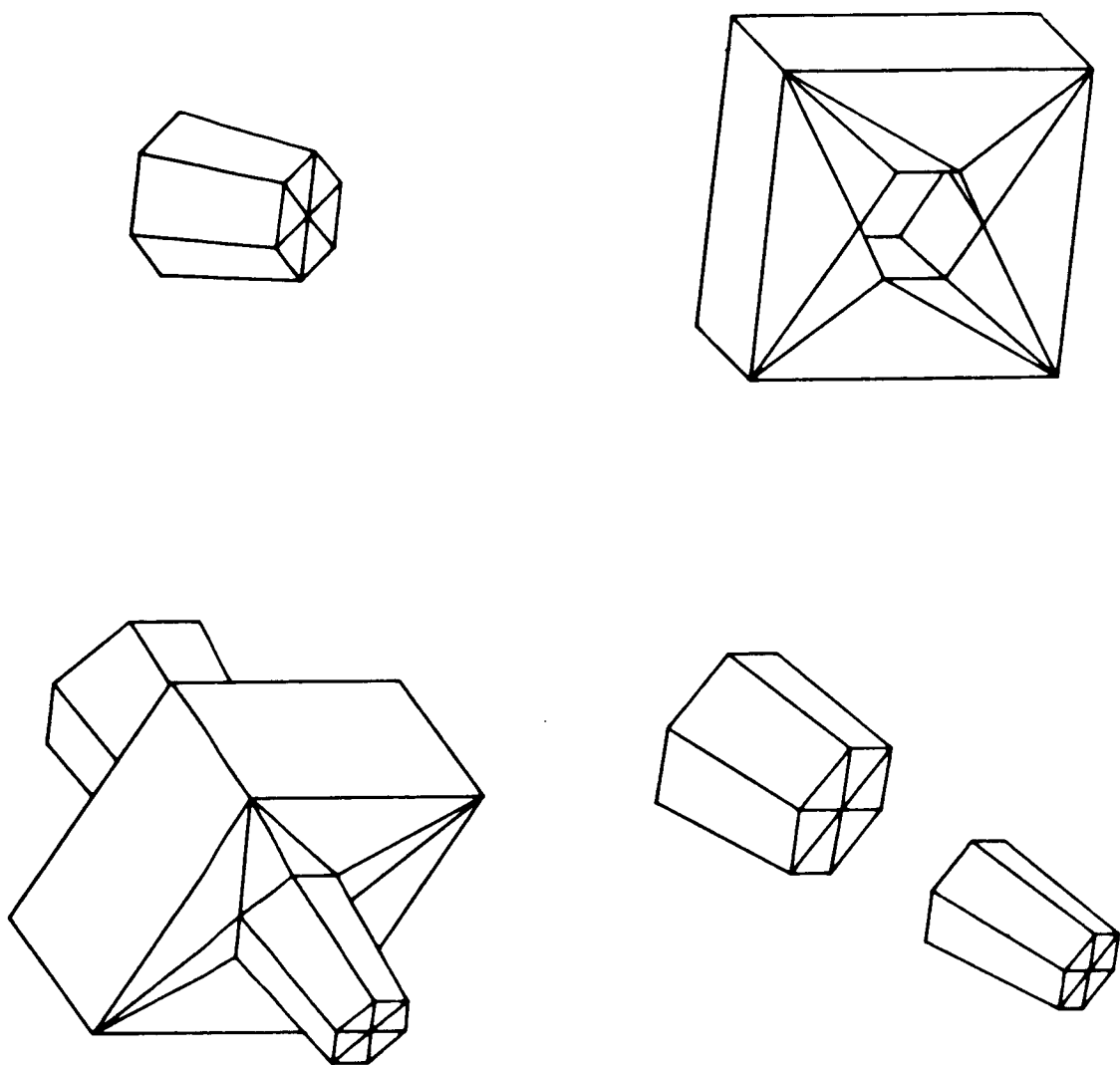


FIGURE 5a
BOOLEAN OPERATION - HIDDEN LINE

ORIGINAL PAGE IS
OF POOR QUALITY

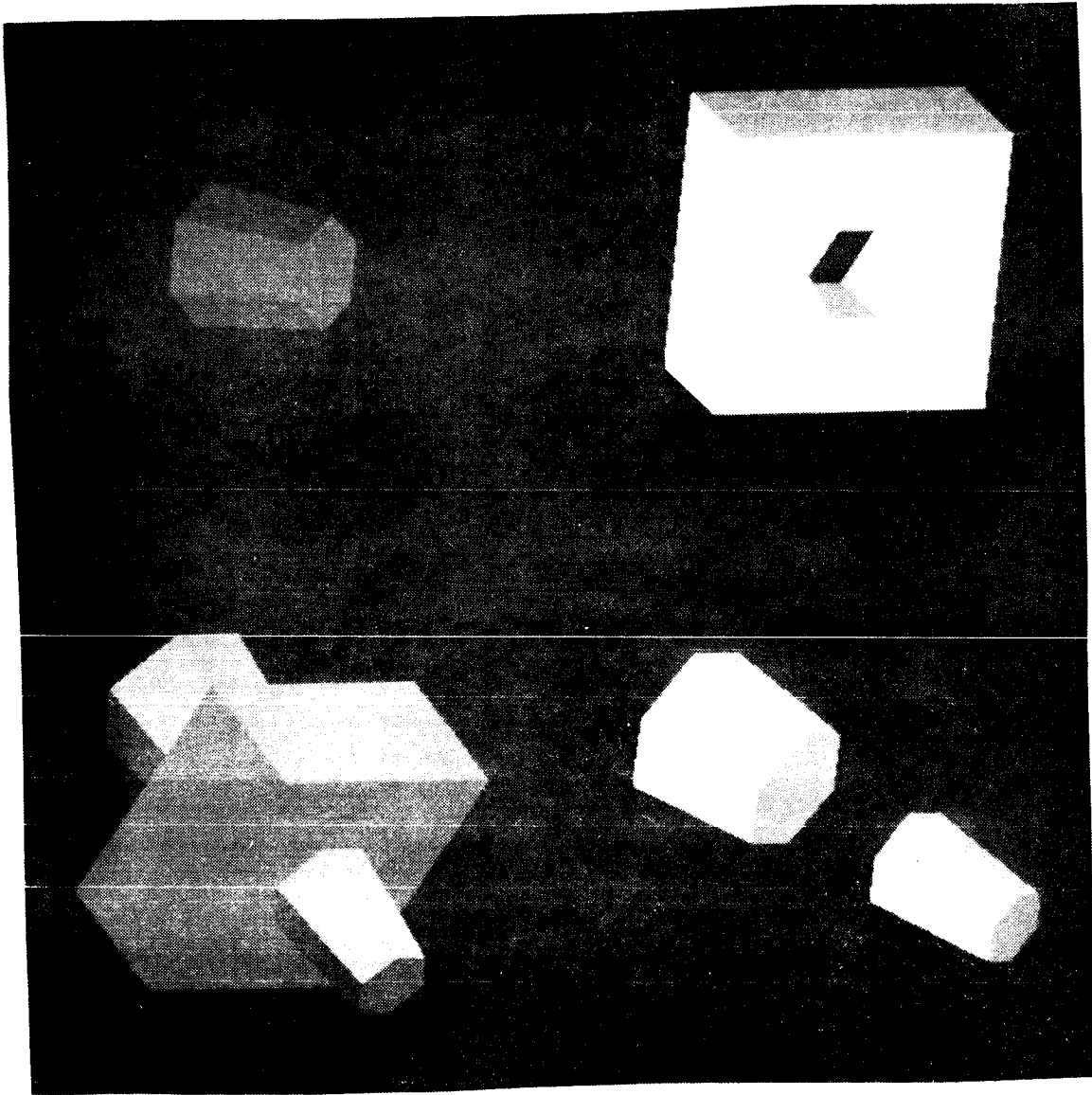


FIGURE 5b
BOOLEAN OPERATION - SHADED IMAGE

If the operands do not geometrically "intersect", a new primitive cannot be constructed. If this occurs, an error message is printed to the user's terminal and all references to this new primitive are implicitly removed from the model.

The algorithm in the current version of the modeller software may encounter computational difficulties when the operand primitives share common "faces", "edges", or "vertices". If the coincident boundary problems cannot be resolved, the error condition is handled in a manner analogous to the non-intersecting primitives.

The Boolean primitive is initially positioned according to the "centroid" of its bounding box; where the "bounding box" is the minimal rectangular hexahedron that contains the primitive. Consequently, the translation parameters in the local transformation matrix are initially set to the coordinates of this centroid. That is, a translation from the "origin" (0,0,0) to the centroid of the bounding box is implicitly performed.

The "intersection" operator could be employed in "static" interference checking where the resultant object shows the extent of the interference.

2.4 EXTERNALS

Solid objects may be composed of irregularly shaped components that cannot be created through "sweeping" (see section 2.2) or repeated applications of the Boolean operators (see section 2.3). In order to accommodate models of this type, an external interface (in the form of an external primitive) is available. The geometry for the irregular primitive is constructed outside of SMP and later used as modeller input.

External parts may be created by an applications program or by a system editor provided they strictly adhere to the format of an SMP geometry file and do not exceed the limitations for a single primitive (see section 3.2).

An external part is exceptional in that it is the only primitive part that is not initially created using the "parts editor" within the modeller (see section 4.3). One (or more) external parts can only be created by reading their respective boundary representations from a formatted "geometry" file (see section 3.2). The initial boundary representation is then stored for future reference. The "centroid" of the external part is defined as the center of the smallest bounding box that contains the complete part. The translation parameters of the local transformation matrix are initialized to the

coordinates of this point. The color is initialized to 7 (white).

After these steps are completed, the external part may be modified using the editors. Modifications, however, are limited to changes in the color, description, and/or local transformation matrix. Should the transformation matrix be changed, the modified matrix will be applied to the initial boundary representation that was stored upon creation. In this way, transformation values are not accumulated.

2.5 ASSEMBLIES

Many solid modeling packages offer a facility for conveniently "grouping" primitives [8]. In SMP the grouping mechanism is called an assembly. Assemblies allow the designer to reference a collection of primitives as a single geometric entity.

The assembly construct is valuable if a collection of primitives is to be replicated one or more times within a model. In conjunction with grouping frequently used primitives, the assembly eases the designer's burden with respect to positioning and orientation. Without the assembly construct, each primitive must be locally transformed in order to achieve proper orientation within the geometric model. With assemblies, the primitive group can be constructed in a convenient location (e.g. the "origin"), and then transformed to the desired orientation with a single local transformation applied to the entire entity.

Assemblies are specified by supplying a list of currently "active" primitives. The present version permits a maximum of 25 primitives (components) to comprise a single assembly. Because assembly components can be selected from the list of all "active" model primitives (including other assemblies),

the above limitation is not too severe. That is, the use of "nested" assemblies is permitted. The other restrictions on assembly definitions are as follows: an assembly must contain at least one component, and an assembly definition must not be "circular." A "circular" assembly definition is one in which an assembly is either directly or indirectly a component of itself.

Assemblies are initially positioned according to the "centroid" of the first component. Therefore, although component order is generally not significant, the user should carefully select the first component. The discussions of the other primitives identify the "centroid" for each primitive type. If the first component happens to be an assembly, then the "centroid" of its first component is utilized. The translation parameters of the local transformation matrix are initialized to the coordinates of the "centroid", implying an initial translation from the origin to the "centroid".

The designation of a single assembly component to initially position the entire assembly is founded on the previous experience the model designer has acquired in transforming primitives. The selection of the first component is arbitrary but appears to be convenient. The "robot arm" model in FIGURE 2 makes extensive use of the assemblies construct, including

the application of nested assemblies. The reader should note that there are at least three local "pivot points" in the model corresponding to movements of the "arm". In each case, assemblies were utilized to depict these areas, and the first component of each assembly represents the "pivot" for each "arm" motion.

If for reasons of necessity or convenience, the user is not satisfied with the "centroid" of any component within the assembly dictating the original transformation, he/she can create an arbitrarily small primitive (e.g. a "box"), position this primitive in the desired location, and force this primitive to be the first component in the assembly. In this manner, the model designer has total control over the initial orientation of assemblies and/or the creation of local "pivots". Because this primitive is arbitrarily small (in relation to the rest of the model), it should have little or no effect on the geometric model as a whole.

The color specified for an assembly will override the color specified for any of its components. That is, all components of the assembly will take on the color of the assembly. If it is desirable to retain the individual component colors, a value of "0" should be entered for the assembly color.

3 GEOMETRIC REPRESENTATION

Solid modeling packages are often distinguished by the manner in which the geometry defining the 3-D object is represented. Because of the importance in selecting the appropriate representation scheme, numerous pure and hybrid schemes have evolved [3]. SMP employs dual geometric representation schemes: a hybrid boundary representation is utilized to manipulate the geometry within the modeller, and a hybrid CSG (constructive solid geometry) scheme serves as the primary user interface.

The motivation for two distinct representation schemes is illustrated in the following example using the "box" (see section 2.1.1) as a typical primitive. If the geometry of the "box" is required for some further computations (e.g. a hidden surface determination), the actual vertices and "connectivity" defining the "edges" and "faces" of the "box" must be made available. For this type of geometry access, a boundary representation scheme is ideal. However, the model designer is not necessarily interested in the 8 vertices, 24 edges, and 6 faces required to represent the boundary of the "box". He/she can unambiguously represent a "box" by applying the three parameters: width, height, and length. A form of a CSG representation is more suited for this type of geometric

reference. Since both types of applications are justifiable, SMP utilizes both representations.

The geometric representation scheme accessible to the user is implemented via the parts-file. The parts-file is a part data base, and contains the information for all "active" primitives required to reconstruct the associated boundary representations. The parts-file includes entries for: part number, part name, part description, dimensions, construction specifications, color, and local transformation. In the case of the composite primitive parts, Booleans (see section 2.3) and assemblies (see section 2.5), information listing the components replaces dimension and construction specifications.

The boundary representation is contained in the so-called geometry-file. Although a formatted geometry-file is available to the user as an input/output option (see section 4.1.2), the geometry-file is a direct access file used for storing the vertices and connectivity for all primitives and their respective components.

The "external" primitive (see section 2.4) is an exception with respect to its geometric representation. Because the "external" part is defined in terms of a boundary representation, this primitive is specified in terms of its

boundary representation on both files.

In the normal mode of operation, there is a one-to-one correspondence between the geometric representation for primitives within the parts-file and the geometry-file. The user has direct control over the contents of the parts-file through the modification of primitive part attributes. Such modifications are reflected in the geometry-file through the reconstruction of the boundary representation for the primitive, thus giving the user indirect control over this file.

The next two subsections will describe the specifics of the two geometric representation schemes.

3.1 PARTS FILE

The user defines a primitive by supplying the minimum amount of information required to construct a boundary representation of the geometric shape and orient it spatially. For each primitive part, this information is entered into a data base, the so-called parts-file. Because a primitive can be unambiguously represented by relatively few attributes, the parts-file information is memory resident during program execution. The actual parts-file is accessed only during the "read" and "write" operations (see sections 4.1 and 4.2). The compactness of the parts-file makes it the ideal mechanism for saving the model during the various stages of creation.

A parts-file comprised of a representative primitive from each of the major categories is shown in FIGURE 6. Examination of this figure illustrates that the layout of the parts-file is primitive dependent. However, the first three records are common to all part types. Regardless of the primitive the first three records are: part identification number, the part name (BOX, CONE, SPHERE, PARABOLOID, TORUS, TRUSS, ROTA-SWEEP, TRANS-SWEEP, BOOLEAN, EXTERNAL, or ASSEMBLY), and part description. The final record for each part, except swept and external parts, is also common to all primitive types. The nine attributes necessary to build the local transformation

matrix reside in this record. The transformation parameters are ordered as follows: x, y, and z rotation angles; x, y, and z scale factors; and the x, y, and z translation values. The differences between the parts-file descriptions for the various primitive types are detailed in the subsequent subsection.

It should be noted that the first record of the parts-file identifies the number of parts to be described.

The user has access to the information in the parts-file via the PRINT command within the EDIT command sublevel (see section 4.3.5).

```

11
1
BOX
THIS IS A CUBE
1.00000E+00 1.50000E+00 1.50000E+00 1.50000E+00 0.00000E-01 0.00000E-01 0.00000E-01 0.00000E-01
0.00000E-01 0.00000E-01 0.00000E-01 1.00000E+00 1.00000E+00 1.00000E+00 -1.00000E+00 0.00000E-01 0.00000E-01
2
SPHERE
THIS IS A MENI-SPHERE
2.00000E+00 1.10000E+00 1.10000E+01 1.00000E+01 1.80000E+02 0.00000E-01 0.00000E-01 0.00000E-01
4.50000E+01 4.50000E+01 4.50000E+01 2.00000E+00 3.00000E+00 4.00000E+00 0.00000E-01 0.00000E-01 0.00000E-01
3
CONE
THIS IS A TRUNCATED CONE
3.00000E+00 2.00000E+00 5.00000E-01 2.00000E+00 3.60000E+02 4.00000E+00 0.00000E-01 0.00000E-01
0.00000E-01 0.00000E-01 0.00000E-01 1.00000E+00 1.00000E+00 1.00000E+00 0.00000E-01 0.00000E-01 0.00000E-01
4
PARABOLOID
THIS IS A PARABOLOID
4.00000E+00 1.00000E+01 1.50000E+00 1.10000E+01 1.00000E+01 3.60000E+02 0.00000E-01 0.00000E-01
0.00000E-01 2.70000E+02 0.00000E-01 1.00000E+00 1.00000E+00 1.00000E+00 1.00000E+01 0.00000E-01 -1.00000E+01
5
TORUS
THIS IS A TORUS
5.00000E+00 1.50000E+00 1.75000E+00 1.00000E+01 1.00000E+01 0.00000E-01 0.00000E-01 0.00000E-01
9.00000E+01 0.00000E-01 9.00000E+01 1.00000E+00 1.00000E+00 1.00000E+00 0.00000E-01 1.00000E+02 0.00000E-01
6
TRUSS
THIS IS A TRUSS
6.00000E+00 5.00000E-01 5.00000E-01 5.00000E-01 2.00000E+00 2.00000E+00 2.00000E+00
2.00000E+00 3.00000E+00 1.00000E+00 0.00000E-01 0.00000E-01 0.00000E-01 0.00000E-01 0.00000E-01 0.00000E-01
0.00000E-01 0.00000E-01 0.00000E-01 1.00000E+00 1.00000E+00 1.00000E+00 0.00000E-01 0.00000E-01 0.00000E-01
7
TRAN-SWEEP
THIS IS A SWEEP STAR
7.00000E+00 5.00000E+00 6.00000E+00 7.00000E+00 1.00000E+00 0.00000E-01 0.00000E-01 0.00000E-01
0.00000E-01 0.00000E-01 0.00000E-01 1.00000E+00 1.00000E+00 1.00000E+00 2.50000E+00 3.00000E+00 3.50000E+00
9
-1.00000E+00 0.00000E-01 0.00000E-01 -2.50000E-01 2.50000E-01 0.00000E-01
0.00000E-01 1.00000E+00 0.00000E-01 2.50000E-01 2.50000E-01 0.00000E-01
1.00000E+00 0.00000E-01 0.00000E-01 2.50000E-01 -2.50000E-01 0.00000E-01
0.00000E-01 -1.00000E+00 0.00000E-01 -2.50000E-01 -2.50000E-01 0.00000E-01
-1.00000E+00 0.00000E-01 0.00000E-01
8
ROTA-SWEEP
THIS IS A SWEEP TORUS
1.00000E+00 1.00000E+01 3.60000E+02 2.00000E+00 0.00000E-01 0.00000E-01 0.00000E-01 0.00000E-01
0.00000E-01 0.00000E-01 0.00000E-01 1.00000E+00 1.00000E+00 1.00000E+00 1.50000E+00 0.00000E-01 -8.34465E-07
5
1.00000E+00 1.00000E+00 0.00000E-01 1.00000E+00 2.00000E+00 0.00000E-01
2.00000E+00 2.00000E+00 0.00000E-01 2.00000E+00 1.00000E+00 0.00000E-01
1.00000E+00 1.00000E+00 0.00000E-01
9
BOOLEAN
THIS IS A BOOLEAN
2.00000E+00
UNION
1 3
0.00000E-01 0.00000E-01 0.00000E-01 1.00000E+00 1.00000E+00 1.00000E+00 -3.75000E-01 -1.19209E-07 -2.38419E-07
10
ASSEMBLY
THIS IS AN ASSEMBLY
3.00000E+00
3 4 5
0.00000E-01 1.80000E+02 0.00000E-01 1.00000E+00 1.00000E+00 1.00000E+00 0.00000E-01 0.00000E-01 0.00000E-01
11
EXTERNAL
7.00000E+00
4.50000E+01 9.00000E+01 1.35000E+02 5.00000E+00 1.00000E+00 1.00000E+00 -1.00000E+00 0.00000E-01 0.00000E-01
8 6
-1.75000E+00 -7.50000E-01 7.50000E-01 -2.50000E-01 -7.50000E-01 7.50000E-01
-2.50000E-01 7.50000E-01 7.50000E-01 -1.75000E+00 7.50000E-01 7.50000E-01
-1.75000E+00 -7.50000E-01 -7.50000E-01 -2.50000E-01 -7.50000E-01 -7.50000E-01
-2.50000E-01 7.50000E-01 -7.50000E-01 -1.75000E+00 7.50000E-01 -7.50000E-01
4 1 2 3 4
4 2 6 7 3
4 5 8 7 6
4 1 4 8 5
4 4 3 7 8
4 1 5 6 2

```

FIGURE 6
PARTS-FILE

3.1.1 BASIC PRIMITIVES PARTS-FILE DESCRIPTION

The basic primitives (see section 2.1) are determined by the appropriate dimension and construction attributes. These attributes are specified in an eight parameter entry which comprises the fourth record of the parts-file for all basic primitives.

TABLE 1 depicts the correspondence between the primitive attributes and the eight parameters for each basic primitive.

TABLE 1A depicts the 16 parameters for the "truss".

TABLE 1
BASIC PRIMITIVE ATTRIBUTES

PARAMETER PRIMITIVE	1	2	3	4	5	6
BOX	color	length	height	width	-	-
CONE	color	positive radius	negative radius	length	revolution angle	number sides
SPHERE	color	radius	number latitudes	number longitudes	revolution angle	-
PARABOLOID	color	length	width	number latitudes	number longitudes	revolution angle
TORUS	color	inner radius	outer radius	number sections	number sides	start angle

TABLE 1A
TRUSS ATTRIBUTES

PARAMETER	1	2	3	4	5	6
	color	element length	element height	element width	horizontal distance between truss elements	vertical distance between truss elements
PARAMETER	7	8	9	10	11	12
	depth dist. between elements	#horiz. bays	#vert. bays	#depth bays	truss top open	truss bottom open
PARAMETER	13	14	15	16		
	truss left open	truss right open	truss front open	truss back open		

3.1.2 SWEPT PART PARTS-FILE DESCRIPTION

The swept parts (see section 2.2) are determined by the appropriate construction attributes and, following the transformation parameters entry, the data defining the 2-D curve to be swept.

The construction attributes are specified in an eight parameter entry which comprises the fourth record of the parts-file. TABLE 2 depicts the correspondence between the attributes and the eight parameters for each swept part. Note that neither of the swept parts require all eight parameter entries.

The data defining the 2-D curve to be swept consists of an entry for the number of vertices entered by the user and the entries containing the coordinates of the vertices. The coordinate entries contain the X, Y, and Z coordinates for each vertex with two vertices per entry. If there is an odd number of vertices then the last entry will contain the X, Y, and Z coordinates of one vertex. The Z coordinate will always be zero because the user defined a 2-D curve in the XY-plane. The Z coordinate was retained in order to maintain consistency with the format of the external part (see section 2.5) and the geometry-file (see section 3.2).

TABLE 2

BASIC SWEPT PART ATTRIBUTES

PARAMETER PRIMITIVE	1	2	3	4	5	6	7
ROTA-SWEEP	color	number sides	revolution angle	end capping	X end cap	Y end cap	-
TRAN-SWEEP	color	X origin	Y origin	Z origin	end capping	X end cap	Y end cap

3.1.3 BOOLEAN PARTS-FILE DESCRIPTION

The Booleans (see section 2.3) are determined by an entry for color, an entry for the operation to be performed, and an entry containing the part identification numbers of the two operands.

3.1.4 EXTERNAL PARTS-FILE DESCRIPTION

The externals (see section 2.4) are determined by an entry for color and a set of entries, following the transformation parameters entry, containing the parts geometry information (see section 3.2). The geometry information consists of an entry for the number of vertices and the number of faces, a set of entries for the coordinates of the vertices, and a set of entries for the connectivity information.

The coordinate entries contain the X, Y, and Z coordinates for each vertex with two vertices per entry. If there is an odd number of vertices then the last entry will contain the X, Y, and Z coordinates of one vertex.

The connectivity entries contain one entry for each face. The entry consists of the number of vertices which define the face (either 3 or 4) and the four indices which reference the vertex entries. The indices are determined by sequentially numbering the vertices such that the Nth vertex will be referenced by the index number N. This scheme of referencing the vertices by one index number rather than the three coordinates (X,Y,Z), saves space by eliminating the redundancy of explicitly stating the (X,Y,Z) coordinates for every occurrence of the vertex. Note that faces with only three

vertices will contain a zero index (i.e. a null reference) in the fourth index position of the connectivity entry.

A comparison of the preceding discussion and TABLE 3 illustrates the close relationship between the external primitive definition and the boundary representation for any primitive.

3.1.5 ASSEMBLY PARTS-FILE DESCRIPTION

The assemblies (see section 2.5) are determined by an entry for color, an entry for the number of components contained in the assembly, and up to three entries for the identification numbers of the components. The component numbers are listed in a ten parameter entry; so for the current maximum of 25 components, three entries would be required.

3.2 GEOMETRY FILE

After user definition of a primitive part, through either the editor "add" command (see section 4.3.1), the "read" command (see section 4.1), or the editor "copy" command (see section 4.3.6), a boundary representation of this part is constructed automatically by SMP. Based on specifications from the part definition (dimension and construction attributes) the boundary representation is generated in a primitive dependent manner (see section 2).

The boundary representations for all parts follow certain conventions. Vertices are determined and connected in such a manner as to compose triangular and/or quadrilateral, planar "faces" which represent (as closely as possible) the boundary of the part. In order to avoid ambiguity, the individual faces are formed by connecting the appropriate vertices in a counter-clockwise order (when viewed from "outside" the object). The number of vertices (except for the "box") is controlled through construction attributes supplied by the user. Increasing the number of vertices (and thus the number of faces) will lead to a closer representation of the true boundary for curved surfaces, but will also increase the complexity (and thus the storage requirements) of the representation. For all primitive parts the maximum number of

vertices is 300 and the maximum number of faces is 600.

In the case of an external part, these restrictions also apply; but planar faces are not mandatory, and the vertex ordering is not restricted. It should be noted, however, that for many features of SMP (e.g. Booleans, back face cull, or hidden surface removal), the results cannot be guaranteed if all of the above conventions are not obeyed.

Internal to modeller software, the boundary representation is then written in binary format to a direct access version of the geometry-file (where each primitive part corresponds to one logical record). When a part is to be displayed or manipulated in other ways, its boundary representation can be quickly retrieved. By this method, only the geometry for a single part need be memory resident.

When a part is modified (a change in its specifications or local transformation matrix), the boundary representation for the part is reconstructed according to its modified attributes and restored in the geometry file.

If the user requires the geometry of his/her model for reasons of further analysis, the boundary representation is available in a formatted version of the geometry-file. This form of the

geometry-file can be generated by invoking the "write" command (see section 4.2). The boundary representation is formatted as shown in TABLE 3.

A sample geometry file entry for a "pyramid" (created with a system editor to be an "external" part) is illustrated in FIGURE 7.

TABLE 3

GEOMETRY FILE DESCRIPTION

RECORD	DESCRIPTION	FORMAT
1	total number of parts on the geometry file	15
(note: the remaining section is repeated for each part)		
2	total number of vertices and total number of faces	215
3	X, Y, Z coordinates of all vertices with 2 vertices per record.	6(1X, 1P, E12.5)
.	(If the number of	
.	vertices is odd, one	
.	additional record will	
N1 (2+(number of vertices/2))	be required for the last vertex.)	
N1+1	descriptions of all faces (one face per record). Each face	515
.	description includes the	
.	number of vertices for	
.	each face and the vertex	
.	indices forming the	
.	face (arranged in a	
.	counterclockwise order	
.	of connectivity). Faces	
.	with only 3 vertices	
N2((N1+1)+(number of faces))	should contain 0 for the fourth vertex.	

1	5	0.0000E-01	-1.0000E+00	-1.0000E+00	-1.0000E+00	0.0000E-01
5	0.0000E-01	-1.0000E+00	-1.0000E+00	1.0000E+00	-1.0000E+00	0.0000E-01
0.0000E-01	-1.0000E+00	-1.0000E+00	1.0000E+00	-1.0000E+00	0.0000E-01	
0.0000E-01	1.0000E+00	0.0000E-01				
0.0000E-01	1	2	0			
5	2	3	0			
5	3	4	0			
5	4	1	0			
5	2	3				
1						

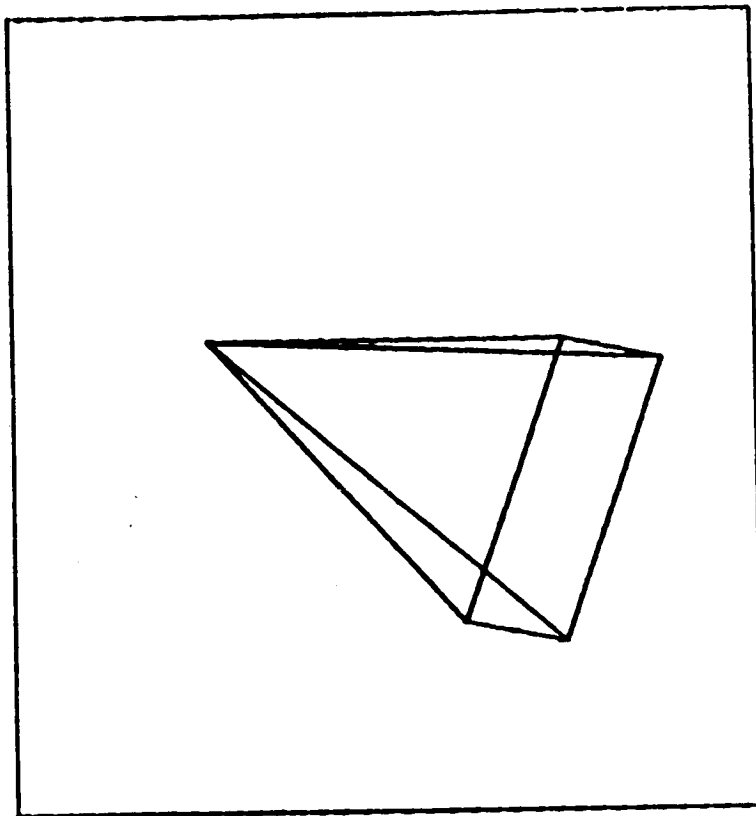


FIGURE 7
GEOMETRY-FILE

4 COMMANDS

SMP provides the designer with a user interface that is both menu and command driven. The user interface can be best understood by examining the hierarchical structure of the modeling system. The individual modeller commands are grouped into levels corresponding to the basic modeling functions.

The "highest" level in SMP is the command level. The available command levels are listed and briefly described below:

READ - input an existing solid model

WRITE - output a solid model

EDIT - modify the solid model by performing
basic editing operations on its primitives

DISPLAY- display the solid model

ANALYSIS- compute mass properties and projected areas
for the solid model

MISCELLANEOUS - perform limited mass properties
analysis and dimensioning on the
solid model

A command level can be comprised of one or more command sublevels. For example, the EDIT command level has a command

sublevel consisting of all available editing operations such as: ADD, MODIFY, COPY, RESTORE, and DELETE. The ADD command sublevel has a second command sublevel menu consisting of the list of available primitives. Once the command sublevels have been exhausted, the individual modeling command may require the user to specify subcommands and command options. Continuing with the above example, if the designer selects the Boolean primitive (see section 2.4), he/she must specify the operands (subcommand) and choose from a set of three operators (command option). The relationships between the various levels is illustrated in TABLE 4.

TABLE 4

SMP COMMAND HIERARCHY

READ (R) EXISTING MODEL

READ PARTS FILE

NEW MODEL / ADD TO CURRENT MODEL
FILE NAME

READ GEOMETRY FILE

NEW MODEL / ADD TO CURRENT MODEL
FILE NAME

READ MOVIE.BYU GEOMETRY FILE

NEW MODEL / ADD TO CURRENT MODEL
FILE NAME

WRITE (W) CURRENT MODEL

WRITE PARTS FILE

SELECT PART RANGE
FILE NAME

WRITE GEOMETRY FILE

SELECT PART RANGE
FILE NAME

WRITE MOVIE.BYU GEOMETRY FILE

SELECT PART RANGE
GEOMETRY FILE NAME
COLOR COMMAND FILE NAME

WRITE PATRAN-G NEUTRAL FILE

SELECT PART RANGE
FILE NAME

EDIT (E) CURRENT MODEL

ADD (A) PART

SUPPLY PART DESCRIPTION

DESIGNATE PRIMITIVE TYPE

BOX
SPHERE

TABLE 4 (cont.)

CONE/CYLINDER
PARABOLOID
TORUS
TRANSLATIONAL SWEEP
ROTATIONAL SWEEP
ASSEMBLY
BOOLEAN
TRUSS

DEFINE ATTRIBUTES (PRIMITIVE DEPENDENT)
DIMENSIONS
CONSTRUCTION PARAMETERS
PROFILE CURVE
COMPONENTS/OPERANDS
BOOLEAN OPERATOR

SPECIFY PART COLOR

MODIFY (M) PART

IDENTIFY PART TO BE MODIFIED

MODIFY OPTIONS
PRINT PART DEFINITION
PRINT PART DEFINITION AND DISPLAY PART
CHANGE PART DESCRIPTION
CHANGE PART SPECIFICATION (PRIMITIVE DEPENDENT)
CHANGE PART TRANSFORMATION
DETERMINE ORDER INDEPENDENT ROTATION ANGLES

DELETE (D) PART

IDENTIFY PART TO BE DELETED
CONFIRM CHOICE

PRINT (P) PART DEFINITIONS

COPY (C) EXISTING PART

IDENTIFY PART TO BE COPIED
CONFIRM CHOICE

RESTORE (R) A DELETED PART

IDENTIFY PART TO BE RESTORED

TABLE 4 (cont.)

CONFIRM CHOICE

DISPLAY (D) THE CURRENT MODEL

RESET (R) VIEWING OPTIONS AND TRANSFORMATION

VIEWING OPTION MENU (M)

SPECIFY PART RANGE

VIEWING TRANSFORMATION OPTION

DEFINE GLOBAL ROTATIONS

DEFINE GLOBAL SCALING

VIEWING OPTIONS

BACK FACE "CULL" OPTION

PART LABELING OPTION

ELEMENT "SHRINKING" OPTION

DRAW (D) THE CURRENT SELECTED PARTS

DISPLAY FOUR VIEWS (T) OF THE CURRENTLY SELECTED PARTS

ZOOM (Z) ON THE CURRENT VIEW

REDRAW OPTION

"PICK" NEW LOWER LEFT CORNER FOR ZOOMED VIEW

"PICK" NEW UPPER RIGHT CORNER FOR ZOOMED VIEW

CONFIRM DIMENSIONS FOR ZOOMED VIEW

HIDDEN SURFACE (S)

HIDDEN SURFACE/LINE OPTION

IMAGE DISPLAY DEFAULT OVERRIDE OPTION

DITHER OPTION

DISPLAY/STORE IMAGE OPTION

RESOLUTION CHANGE OPTION

SHADING OPTION

FLAT

SMOOTH

SHADING PARAMETER OPTION

DIFFUSED LIGHT VALUE

REGULAR LIGHT EXPONENT

BACK FACE "CULL" OPTION

COMPOSITE HIDDEN SURFACE-FOUR VIEW (C)

(SEE HIDDEN SURFACE (S) COMMAND)

TABLE 4 (cont.)

GRAPHICS (G) EDITOR

PICK A PART (P)

SPECIFY XY, YZ, OR ZX-PLANE VIEW

PICK PART CENTER (LABEL)

(SEE DISPLAY MENU COMMAND)

PICK AND MODIFY (M) A PART

(SEE GRAPHICS EDITOR PICK A PART COMMAND)

(SEE EDIT MODIFY COMMAND)

PICK AND DELETE (D) A PART

(SEE GRAPHICS EDITOR PICK A PART COMMAND)

(SEE EDIT DELETE COMMAND)

PICK AND COPY (C) AN EXISTING PART

(SEE GRAPHICS EDITOR PICK A PART COMMAND)

(SEE EDIT COPY COMMAND)

PICK AND PRINT COORDINATES (N) OF A POINT

(SEE GRAPHICS EDITOR PICK A PART COMMAND)

(PICK POINT WHOSE COORDINATES ARE DESIRED)

TRANSLATE (T) A PART

TRANSLATION OPTION

TRANSLATE BY PART CENTROID

(SEE GRAPHICS EDITOR PICK A PART COMMAND)

TRANSLATE BY PART VERTEX

(SEE GRAPHICS EDITOR PICK A POINT COMMAND)

PICK NEW/EXISTING TRANSLATION REFERENCE POINT

CONFIRM TRANSLATION SELECTIONS

ROTATE (R) A PART

TABLE 4 (cont.)

(SEE GRAPHICS EDITOR PICK A PART COMMAND)
PICK ROTATION REFERENCE POINT
RELOCATE ROTATION REFERENCE POINT
CONFIRM ROTATION SELECTIONS

ANALYSIS (A)

MASS PROPERTIES (P)

SPECIFY PARTS RANGE
VIEWING TRANSFORMATION OPTION
 DEFINE GLOBAL ROTATIONS
 DEFINE GLOBAL SCALING
COMPUTATION GRID SIZES DEFAULT OVERRIDE OPTION
 DEFER COMPUTATIONS TO BATCH MODE

PROJECTED AREAS (A)

SPECIFY PARTS RANGE
VIEWING TRANSFORMATION OPTION
 DEFINE GLOBAL ROTATIONS
 DEFINE GLOBAL SCALING
COMPUTATION GRID SIZE DEFAULT OVERRIDE OPTION
 DEFER COMPUTATIONS TO BATCH MODE

MISCELLANEOUS (P) COMMANDS

PART DIMENSIONS (D)

DIMENSIONING OPTION
 DISTANCE BETWEEN PART CENTROIDS
 DISTANCE BETWEEN NODE POINTS
 DISTANCES BETWEEN ALL CENTROIDS
(SEE GRAPHICS EDITOR PICK A PART COMMAND)

GVS (G)

MODIFY DENSITIES (D)

MODIFY DENSITIES
 PRINT
 MODIFY

TABLE 4 (cont.)

PART ID/DENSITY
MODIFY MASSES
PRINT
MODIFY
PART ID/MASS

MASS PROPERTIES (P)
(SEE DISPLAY MENU COMMAND)
(SEE ANALYSIS MASS PROPERTIES COMMAND)

AREAS AND BLOCKAGES (A)
(SEE ANALYSIS PROJECTED AREAS COMMAND)
DEFER COMPUTATION TO BATCH MODE

LOCATE ADDED NODES (L)

ADD A NODE

DISPLAY THRUSTERS/PROPELLANT TANKS/AMCD'S

LOCATE BY KNOWN COORDINATES

ENTER NODE ID

PRINT ALL NODES

LOCATE NODE GRAPHICALLY

(SEE DISPLAY MENU COMMAND)

DISPLAY THRUSTERS/PROP TANKS/AMCD'S/

ALL NODES/ONLY CURRENT NODE

ASSOC. ADDED NODE W/EXISTING S/C PART

COORDINATE VIEW PLANE OPTION

ADD NEW NODE

IDENTIFY EXISTING POINT

IDENTIFY NEW POINT

DELETE A NODE

IDENTIFY NODE BY NODE ID

ENTER NODE ID

PRINT ALL NODES

IDENTIFY NODE GRAPHICALLY

(NOT IMPLEMENTED)

TABLE 4 (cont.)

MODIFY A NODE

IDENTIFY NODE BY NODE ID

ENTER NODE ID

PRINT ALL NODES

IDENTIFY NODE GRAPHICALLY

(SEE GVS ADD A NODE COMMAND)

DISPLAY ADDED NODES

PRINT NODE DESCRIPTIONS

(SEE GVS ADD A NODE - LOCATE NODE

GRAPHICALLY COMMAND)

INITIALIZE GVS (I)

UNITS CONVERSION FACTOR DEFAULT OVERRIDE OPTION

WRITE GVS OPTION FILE (W)

MASS PROPERTY/PROJECTED AREA FILE

GVS COMMON FILE NAME

MASS PROPERTY/PROJECTED AREA FILE AND

ARCD INPUT DATA TO RIM DB

GVS COMMON FILE NAME

READ GVS COMMON DATA FILE (R)

GVS COMMON FILE NAME

WRITE GVS BATCH COMMON FILE (B)

GVS COMMON FILE NAME

Although the formal level structure of the SMP commands may appear unnecessarily complex, the procedures for user input remain essentially identical regardless of whether the input refers to a command level, command sublevel, subcommand, or command option. For instance, the appropriate command level and command sublevels are designated by entering the first character of the level name (e.g. "E" for EDIT). If an illegal character is encountered, the user is prompted with a list of legal responses and then permitted to re-enter the selection. In the case of the subcommand and the command option, the user is first prompted for a response. The input for these levels is command dependent; but in all cases, an illegal response will result in the prompt being redisplayed. Input at all SMP levels is "free field".

The program prompts often serve to identify the current command level or subcommand level. For example, the command level is identified by the prompt

ENTER COMMAND>.

The DISPLAY and EDIT command sublevels can be distinguished by the two prompts

ENTER DISPLAY COMMAND>

or

ENTER EDITOR COMMAND>.

There are certain features common to all command levels and command sublevels. These features will be discussed in the following paragraphs.

An on-line "help" facility is available for all command levels. The "help" (HELP) feature permits the user to examine selected sections of this document at his/her interactive terminal during SMP execution. HELP is invoked by entering the character "H" from the command or command sublevel. General information concerning the function of the command level is automatically displayed, and the user can opt for more detailed information regarding the lower levels.

The user can normally traverse back to the next highest level by entering "Q" (for "QUIT"). QUIT works at all command levels, most command sublevels, and at designated lower levels. Entering "Q" from the command level causes an "exit" from the program. If one or more primitives have been modified since the parts-file (see section 3.1) has been last "replaced", the user is prompted regarding the permanence of these modifications.

If the user desires to exit the system without traversing to the upper command levels, the "immediate exit" ("X") option should be used. The "exit" option can be invoked from any

command level and most command sublevels. Naturally, the use of the "X" option implies that modifications made to the model since the last "replace" are ignored.

SMP has a built-in interrupt capability which allows the user to stop the execution of a given process and to return to the current menu level. The interrupt is somewhat "host-dependent" (see section 6.1.2). After the "interrupt" key has been struck, the screen will be erased and the current menu level prompt will be issued. This feature is most useful in terminating the drawing of a model that requires corrections, without forcing the user to wait until the display is completed.

Several commands require the user to indicate the primitives to be included in a particular operation. If the model is comprised of a large number of parts, the user may have occasion to view only selected parts as a time saving measure. The selection procedure provides options for the complete model ("*"), part ranges (delimited by a "-"), and individual parts (delimited by a ","). For example, consider the model comprised of 15 primitive parts. The input line:

2,4,6-9,11-13,15

results in parts 2,4,6,7,8,9,11,12,13, and 15 being included in this operation. "Deleted" parts (see section 4.3.3) and

"component" parts (see sections 2.3 and 2.5) are exceptions to the above convention. Deleted parts are never included in a specified operation. Component parts may be included or excluded depending on the nature of the operation. If parts 4 and 8 were deleted in the earlier example, the identical input line includes only the parts 2,6,7,9,11,12,13, and 15.

The notion of a "part status" has been alluded to but never explained. A primitive can be either active or inactive. When a part is initially created, its status is "active"; and it remains "active" until one of two conditions exists: the part is explicitly or implicitly deleted, or the part becomes a component of a Boolean or assembly. (The rationale behind this second condition is that assembly components and Boolean operands are building tools that can be discarded once the composite part is constructed.) Because the status of a part may effect the result of an SMP command, the user should be alerted as to which commands are impacted by part status. An "inactive" status can be changed to "active" via the RESTORE command within the EDIT command level (see section 4.3.4).

4.1 READ COMMAND LEVEL

The READ (R) command serves as both a means of inputting parts to SMP that have previously been "saved" (see section 4.2) and as an interface to other programs.

Currently, three types of files may be read: a parts-file (see section 3.1), a geometry-file (see section 3.2), or the input geometry from the MOVIE.BYU graphics system [9]. Reading a parts-file is the only means of retaining a part's complete identity; whereas geometry information read from either of the other files is used to create only "external" parts (see section 2.4). The "editing" restrictions applying to "external" parts have been previously discussed (see sections 2.4 and 3.2).

After the file type is specified, a choice is given to either destroy the existing model and make a new model consisting only of the parts contained in the new parts-file, or to append the new parts to the existing model. By employing the append capability new models can be constructed by reading and combining several part-files. This can be convenient for applications where models are composed of similar components.

Finally, a file name must be specified. If the file does not

exist or cannot be successfully "opened", the user is informed and asked for another file name. If an error occurs in the attempt to read the file, the current model remains unchanged.

Part numbering ambiguities can result if parts were deleted in an earlier session or if parts-files are being concatenated. One of the functions of the READ operation is to resolve such difficulties by assigning a unique number to each part.

READ also flags each part as "active" or "inactive". All parts are tagged as "active" with the exception of parts which are components of composite parts (i.e. Booleans and assemblies).

4.2 WRITE COMMAND LEVEL

The WRITE (W) command allows a user to save selected parts of the current model at any time during the modeling process. Employing this capability, the user may interrupt a modeling session and continue at a later time, or save designated model parts in separate files for use in other models. The WRITE command also provides a link between SMP and other software systems.

Currently, parts may be saved in four different formats: a parts-file (see section 3.1), a geometry-file (see section 3.2), a MOVIE.BYU [9] geometry input file, or a PATRAN-G [10] "neutral" file.

After specifying the file type, the individual parts to be saved are selected (see section 4). For the parts-file, all parts that are explicitly named are saved unless they have been "deleted" (see section 4.3.4). Also, all components of explicitly named composite parts are implicitly saved. Part I.D. numbers remain unchanged on the saved parts-file, but are renumbered consecutively when the file is subsequently "read" (see section 4.1) by SMP. For all three geometry files, explicitly named parts are saved only if they are currently "active" (see section 4). (Hence, component parts

are not saved unless explicitly "restored" (see section 4.3.4)). In the case of MOVIE.BYU output, the designer can opt to store all components of an "assembly" as a single "MOVIE.BYU part" or to store each assembly component as an individual "part".

Finally, a file name is requested for the saved model. If the file already exists, a choice is given to either overwrite the existing file or input an alternate file name.

For the MOVIE.BYU file option only, a second file may be written containing the current color values for the saved parts. The file is formatted as a MOVIE.BYU "command" file and contains input commands to specify part colors for the MOVIE.BYU DISPLAY program. If all components of an assembly are to become one MOVIE part, the color of that part remains the same as that of the assembly (white for assembly color 0). If each component is to become a single MOVIE part, all will retain their component colors. The use of this file alleviates the need for the user to have to re-enter the part colors specified in SMP within the MOVIE.BYU system. The user is prompted for this second file name. Again, a choice is given to overwrite the file, if it exists, or to specify an alternate name.

For the PATRAN file option, each facet of the SMP geometric representation is converted to a PATRAN "quad" element type. The resultant PATRAN model may contain less efficient representations of primitives than could be achieved using PATRAN alone.

4.3 EDIT COMMAND LEVEL

The commands in the EDIT (E) command level permit the designer to either create a new geometric model through the composition of primitives, or to "edit" an existing geometric model through the addition of new primitives or modification of existing primitives. The primitive editing functions include:

- ADD
- MODIFY
- DELETE
- RESTORE
- PRINT
- COPY

The editing commands perform an operation synonymous with the command name. The detailed operation of each of the editing functions is described in a subsequent subsection. The invocation of an EDIT command results in an explicit action being performed on the designated part. However, if the geometric model is comprised of composite parts (either assemblies or Booleans), the editing operation may result in implicit modifications of other composite parts. For example, the explicit deletion of a primitive which is also an assembly component results in the implicit removal of this primitive from the assembly component list. If the composite parts are "nested", an operation on a component can have far reaching effect on the model as a whole. It should be noted that the RESTORE command (see section 4.3.4) prevents operations such

as the one described above from being irrevocable.

The preceding discussion presumes the user is working with the parts-file representation (see section 3.1) of the geometric model. In this case, only the geometry-file representation (see section 3.2) of the model is available, some editing functions are illegal. Recall that in a geometry-file representation all parts are treated as externals (see section 2.4). Therefore, editing functions prohibited for external primitives are also prohibited for geometric file entries. More specifically, the user can alter the local transformation parameters, but not the actual part geometry.

4.3.1 ADD

The ADD (A) command sublevel allows the user to create a new part. The added part may be a basic primitive, a swept part, an assembly, or a Boolean part, but not an external part (see section 2.4).

A list of the available primitives is printed; and the user enters the letter which corresponds with the desired primitive, or a "Q" to return to the EDITOR level:

```
ENTER PART NAME - BOX(B), SPHERE(S), CONE(C)
                  PARABOLOID(P), TORUS(T), TRUSS(U), TRANSLATIONAL-SWEEP(N)
                  ROTATION-SWEEP(R), ASSEMBLY(A), BOOLEAN(E)
                  OR QUIT(Q)
```

After a legal selection, the user is requested to enter the part description (maximum of 80 characters) or a carriage return for no description.

Following the description is a series of requests for the dimension and construction attributes of the selected primitive. These attribute requests are primitive dependent, and the order of these requests is mirrored in the preceding primitive discussions (see sections 2.1-2.3 and 2.5). The final part attribute to be entered is the color code (see section 2). If any of the attributes are not within the specified range (again primitive dependent); a warning

message is printed specifying the problem, and the corrective action is taken in overcoming the problem.

The local transformation values are defaulted to no rotations, unit scaling, and no translations (except for the non-basic primitive parts which are not centered at the origin (see section 2)).

The user is automatically transferred to the MODIFY command sublevel (see section 4.3.2) where attributes can be edited and/or the primitive reoriented.

4.3.2 MODIFY

The MODIFY (M) command sublevel provides a means of changing the attributes of a "non-deleted" (see section 4.3.3) part. The attributes that can be changed include the dimension parameters, construction parameters, orientation parameters, part description, and the color. The MODIFY sublevel can be reached either directly by the MODIFY command, or indirectly by the ADD (see section 4.3.1) or COPY (see section 4.3.6) sublevel commands.

The explicit modification of a component part causes the implicit modification of all composite parts containing this component.

After issuing the MODIFY command, the user is requested to enter the part identification number of the part to be modified or a QUIT (or Q) to return to the EDITOR command level. If an invalid part number is entered (i.e. the part is deleted or non-existent), the request is repeated.

A menu of eight options is displayed next, and the user must enter an integer between 1 and 8 which corresponds with the desired option:

1 PRINT PART DEFINITION

- 2 PRINT PART DEFINITION AND DISPLAY PART
- 3 CHANGE PART DESCRIPTION
- 4 CHANGE PART SPECIFICATION
- 5 CHANGE PART TRANSFORMATION
- 6 DETERMINE ORDER INDEPENDENT ROTATION ANGLES
- 7 CHANGE PART COLOR
- 8 TO RETURN TO EDITOR (PART OK)

The first two options allow the user to verify that the selected part I.D. number matches the part to be modified and that the most recent modifications are correct by printing the current part definition (see section 4.3.5) or by printing the current part definition and displaying the part (see section 4.4.2). The third option allows the user to enter a new part description (maximum of 80 characters).

The fourth option allows the user to change the dimension and construction parameters of the selected part. The method for changing the parameters varies depending on the part type. Note that the geometry of an external part cannot be modified in this manner (see section 2.4). The part definition is printed to provide a reference for the user.

If the part is a basic primitive, the user is requested to enter an attribute number followed by a new value for each change. The attribute number is the column number which is aligned with the attribute descriptions and values in the printed (see section 4.3.5) part definition. The column number is therefore used as an index for the attribute value.

The attribute number must be between 2 and 9; attribute number 1 is the part identification number which cannot be changed by the user. For instance, if the selected part is a "box", the entry

3,32

changes the height of the box to 32.0 (see TABLE 1). When the user has completed all part attribute changes, he/she terminates by entering 0,0. The changes are then made, and the MODIFY menu is redisplayed.

If the part is a swept primitive then a second submenu is displayed:

- 1 CHANGE INPUT POINT COORDINATES
- 2 INSERT A NEW POINT
- 3 DELETE AN EXISTING POINT
- 4 CHANGE PART SPECIFICATION
- 5 RETURN

The first three suboptions allow the user to modify the (X,Y) coordinates that define the 2-D curve (see section 2.2). Each (X,Y) coordinate is assigned an index number; so the user simply enters the index number followed by the new X and Y values or a (0,0,0) to terminate. For the first suboption, the user specified index number must be an existing index number so that the old (X,Y) coordinates can be replaced. For example, if the first suboption is chosen, the entry

3,12,5,15.

changes the coordinates of the third point to

(12.5,15.0).

For the second suboption, the new coordinates will be inserted after the specified index number. For the third suboption, the (X,Y) coordinates are not needed since the coordinates referenced by the index number will be deleted. The fourth suboption is identical to the construction specification modifications used for the basic primitives. The fifth suboption returns directly to the MODIFY menu without making any changes; the first four options return to the MODIFY menu upon normal completion of the operation.

If the part is an assembly, another submenu is displayed:

- 1 RE-INPUT COMPONENT ARRAY
- 2 INSERT A NEW COMPONENT
- 3 DELETE AN EXISTING COMPONENT
- 4 RETURN

The first suboption allows the user to re-enter the entire component array by entering one component number per line and terminating with a "Q".

For example, if the first suboption is chosen, the entries:

9
2
5
Q

replace the old component array with the ordered components 9, 2, and 5.

The second suboption either appends the new component to the end of the component array or else "re-activates" the component in its previous location if it had been deleted in a previous session. The third suboption makes an existing component inactive. Care should be taken when modifying the component array since the orientation of the assembly is based on the first component. The second and third suboptions notify the user if the first component is modified and provide a query to allow the user to recover if this is an undesired result. The fourth suboption returns to the MODIFY menu as do the other three options when their actions have been completed.

If the part is a Boolean part, another submenu is displayed:

- 1 CHANGE OPERATOR
- 2 CHANGE OPERANDS
- 3 RETURN

Suboptions one and two provide the same prompts as the ADD sublevel command (see section 4.3.1) and return to the MODIFY menu after completion. The third option returns directly to the MODIFY menu without changing the part.

The fifth option on the MODIFY menu allows the user to modify the orientation parameters. The same indexing technique, as described for the basic primitive attribute modifications, is

used to modify the transformation parameters. The nine indices reference the X, Y, and Z rotations, scale factors, and translations. For example, the entries

2,90
4,10
9,5.5
0,0

change the Y rotation angle to 90.0 degrees, the X scale factor to 10.0, and the Z translation to 5.5.

The sixth option gives the user a means of entering any number of rotation angles and in any order. The result is the equivalent order dependent X, Y, and Z rotations. The user alternates between entering the axis for the rotation and the angle value in degrees. When the user is finished, a "Q" for quit is entered which causes the rotation values to be replaced with the newly calculated order dependent X, Y, and Z rotation angles.

The seventh option allows the user to change the color of the selected part by entering a color code value between 1 and 7.

The eighth option returns to the EDITOR command level when all of the part modifications are complete.

4.3.3 DELETE

The DELETE (D) command allows the user to tag a part as inactive. A "deleted" part cannot be displayed, modified, used as a component, or written to a file unless the part is made active again by issuing the RESTORE command (see section 4.3.4). It should be noted that all "deleted" parts are permanently removed when the user saves (see section 4.2) the geometric model.

After issuing the DELETE command, the user is requested to enter the part identification number of the part to be deleted or a QUIT (or Q) to return to the EDITOR command level. If an invalid part number is entered (i.e. the part is already deleted or non-existent), the request is repeated.

A menu of five options is displayed next, and the user must enter an integer between 1 and 5 which corresponds with the desired option:

- 1 TO PRINT PART DESCRIPTION
- 2 TO PRINT PART DESCRIPTION AND DISPLAY PART
- 3 TO CONFIRM DELETION
- 4 TO MAKE NEW SELECTON
- 5 TO RETURN TO THE EDITOR

The first two options allow the user to verify that the selected part I.D. number matches the part to be deleted by printing the part definition (see section 4.3.5) or by

printing the part definition (see section 4.3.5) and displaying the part (see section 4.4.2). The third option allows the user to confirm the deletion and return to the EDITOR command level. The fourth option allows the user to select a different part for deletion, and the fifth option returns to the EDITOR command level without deleting the part.

By deleting a part (explicit deletion), one or more other parts may also be deleted (implicit deletion). The explicit deletion deletes the selected part; and if the part is a component of other composite parts, those components are also implicitly removed. An implicit deletion occurs if the deleted part is the only component of an assembly or one of the operands of a Boolean part. The user is notified when an implicit deletion occurs so that corrective action (see section 4.3.4) could be taken if the implicit deletion was accidental.

If the first component of an assembly is deleted by either an explicit or an implicit part deletion, a warning message and a query will be issued at the first occurrence of this condition. The user can recover by entering "N" to ignore the deletion and return to the editor. By entering "Y", the deletion of the first component of this assembly and all subsequent assemblies in which this component is the first

component will be performed. The deletion of the first component of an assembly warrants this level of caution since the first component affects the positioning and orientation of the assembly (see section 2.5).

4.3.4 RESTORE

The RESTORE (R) sublevel command allows the user to re-activate an inactive part. By using the RESTORE command, a part which was deleted in a given session can be re-established as an available part during the same session, and a component part can be returned to active part status.

After issuing the RESTORE command, the user is requested to enter the part identification number of the part to be restored or a QUIT (Q) to return to the EDITOR command level. If an invalid part number is entered (i.e. the part is already active or non-existent), the request is repeated.

A menu of five options is displayed next, and the user must enter an integer between 1 and 5 which corresponds to the desired option:

- 1 TO PRINT PART DESCRIPTION
- 2 TO PRINT PART DESCRIPTION AND DISPLAY PART
- 3 TO CONFIRM RESTORATION
- 4 TO MAKE NEW SELECTION
- 5 TO RETURN TO THE EDITOR

The first two options allow the user to verify that the selected part I.D. number matches the part to be restored by printing the part definition (see section 4.3.5) or by printing the part definition and displaying the part (see section 4.4.2). The third option allows the user to confirm

the restoration and return to the EDITOR command level. The fourth option allows the user to select a different part for restoration, and the fifth option returns to the EDITOR command level without restoring the part.

The restoration of a non-deleted part which is a component of one or more composite parts causes three changes in the part's status. The three changes are: the part can be displayed, the printing of the part description no longer contains the "component part" message, and the part can be written to a geometry file.

The restoration of a previously deleted part causes the part to be made active again; and if the part is a component of a composite part, the component is also reactivated. It should be noted that unlike the DELETE command, no implicit restorations occur (see section 4.3.3). Consequently, a RESTORE command must be issued for the originally deleted part and each implicitly deleted part when the user wants to undo a previous DELETE command which caused implicit deletions.

If the first component of an assembly is restored by the restoration of a previously deleted part, a warning message and a query will be issued at the first occurrence of this condition. The user can recover by entering "N" to ignore the

restoration and return to the editor. By entering "Y", the restoration of the first component and all subsequent first component restorations will be performed. The restoration of the first component of an assembly warrants this level of caution since the first component affects the positioning and orientation of the assembly (see section 2.5).

4.3.5 PRINT

The PRINT (P) command sublevel prints the descriptive information for all parts that have not been "deleted". The descriptive information includes the identification number, the color, the dimension and construction parameters, the orientation parameters, and the part description.

The print format is illustrated in FIGURE 8 where a representative from each part "type" is included. (As a space saving measure the "types" are not separated by pages.) The order used in this figure parallels the order in PRINT.

The part "type" is written at the top of the display area followed by a header. The header consists of a row of column numbers, a row of part attribute descriptions, and a row of transformation descriptions. The descriptive information for one or more parts follows the header. If a part is a component of a composite part, the message:

COMPONENT PART

is appended to the part's descriptive information.

To signal the user that output to the display area is complete, the terminal's "bell" is rung and a message is written informing the user to enter a carriage return to

advance to the next "page" of information. The user is returned to the EDITOR command level at the completion of the print output.

It should be noted that the format used in the PRINT command is maintained in all of the EDITOR sublevel commands when printing a part's description.

ASSEMBLIES

PART ID	10	COLOR	3						
COMPONENTS	3	4	5						
1	2	3	4	5	6	7	8	9	
ROTATEX(DEG)	ROTATEY(DEG)	ROTATEZ(DEG)	SCALEX	SCALEY	SCALEZ	TRANSLATEX	TRANSLATEY	TRANSLATEZ	
0.00000E-01	1.80000E+02	0.00000E-01	1.00000E+00	1.00000E+00	1.00000E+00	0.00000E-01	0.00000E-01	0.00000E-01	
THIS IS AN ASSEMBLY									

BOXES

PART ID	2	3	4	5	6	7	8	9
ROTATEX(DEG)	ROTATEY(DEG)	LENGTH(X)	HEIGHT(Y)	WIDTH(Z)	SCALEZ	TRANSLATEX	TRANSLATEY	TRANSLATEZ
1	1	1.50000E+00	1.50000E+00	1.50000E+00	1.00000E+00	-1.00000E+00	0.00000E-01	0.00000E-01
0.00000E-01	0.00000E-01	0.00000E-01	1.00000E+00	1.00000E+00	1.00000E+00	-1.00000E+00	0.00000E-01	0.00000E-01
THIS IS A CUBE								
*** COMPONENT PART ***								

SPHERES

PART ID	2	3	4	5	6	7	8	9
ROTATEX(DEG)	ROTATEY(DEG)	RADIUS	#LATITUDES	#LONGITUDES	REV. ANGLE	TRANSLATEX	TRANSLATEY	TRANSLATEZ
2	2	1.10000E+00	11	10	1.80000E+02	0.00000E-01	0.00000E-01	0.00000E-01
4.50000E+01	4.50000E+01	4.50000E+01	2.00000E+00	3.00000E+00	4.00000E+00	0.00000E-01	0.00000E-01	0.00000E-01
THIS IS A HEMI-SPHERE								

CONES

PART ID	2	3	4	5	6	7	8	9
ROTATEX(DEG)	ROTATEY(DEG)	POS. RADIUS	NEG. RADIUS	LENGTH	REV. ANGLE	#SIDES	TRANSLATEX	TRANSLATEY
3	3	2.00000E+00	5.00000E-01	2.00000E+00	3.60000E+02	4	0.00000E-01	0.00000E-01
0.00000E-01	0.00000E-01	0.00000E-01	1.00000E+00	1.00000E+00	1.00000E+00	0.00000E-01	0.00000E-01	0.00000E-01
THIS IS A TRUNCATED CONE								
*** COMPONENT PART ***								

PARABOLOIDS

PART ID	2	3	4	5	6	7	8	9
ROTATEX(DEG)	ROTATEY(DEG)	LENGTH(X)	WIDTH(Y)	#LATITUDES	#LONGITUDES	REV. ANGLE	TRANSLATEX	TRANSLATEY
4	4	1.00000E+01	1.50000E+00	11	10	3.60000E+02	0.00000E-01	-1.00000E+01
0.00000E-01	2.70000E+02	0.00000E-01	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+01	0.00000E-01	-1.00000E+01
THIS IS A PARABOLOID								
*** COMPONENT PART ***								

TORI

PART ID	2	3	4	5	6	7	8	9
ROTATEX(DEG)	ROTATEY(DEG)	INNER RADIUS	OUTER RADIUS	#SECTIONS	#SIDES	START ANGLE	TRANSLATEX	TRANSLATEY
5	5	1.50000E+00	1.75000E+00	10	10	0.00000E-01	0.00000E-01	1.00000E+02
9.00000E+01	0.00000E-01	9.00000E+01	1.00000E+00	1.00000E+00	1.00000E+00	0.00000E-01	1.00000E+02	0.00000E-01
THIS IS A TORUS								
*** COMPONENT PART ***								

FIGURE 8
EDITOR PRINT FORMAT

EXTERNAL

1 PART ID	2 COLOR	3 ROTATEX(DEG)	4 ROTATEY(DEG)	5 ROTATEZ(DEG)	6 SCALEX	7 SCALEY	8 SCALEZ	9 TRANSLATEX	10 TRANSLATEY	11 TRANSLATEZ
11	7	4.50000E+01	9.00000E+01	1.35000E+02	5.00000E+00	1.00000E+00	1.00000E+00	-1.00000E+00	0.00000E-01	0.00000E-01

TRAN-SWEEPS

1 PART ID	2 COLOR	3 ORIGINX	4 ORIGINY	5 ORIGINZ	6 CAPS (1=YES)	7 CAP (X)	8 CAP (Y)	9 CAP (Z)	10 TRANSLATEX	11 TRANSLATEY	12 TRANSLATEZ
7	7	5.00000E+00	6.00000E+00	7.00000E+00	1	0.00000E-01	0.00000E-01	0.00000E-01	2.50000E+00	3.00000E+00	3.50000E+00

THIS IS A SWEEP STAR

POINT 1 (X,Y) : -1.00000E+00 0.00000E-01
 POINT 2 (X,Y) : -2.50000E-01 2.50000E-01
 POINT 3 (X,Y) : 0.00000E-01 1.00000E+00
 POINT 4 (X,Y) : 2.50000E-01 2.50000E-01
 POINT 5 (X,Y) : 1.00000E+00 0.00000E-01
 POINT 6 (X,Y) : 2.50000E-01 -2.50000E-01
 POINT 7 (X,Y) : 0.00000E-01 -1.00000E+00
 POINT 8 (X,Y) : -2.50000E-01 -2.50000E-01
 POINT 9 (X,Y) : -1.00000E+00 0.00000E-01

ROTA-SWEEPS

1 PART ID	2 COLOR	3 #SIDES	4 REV. ANGLE	5 CAPS (1=YES)	6 CAP (X)	7 CAP (Y)	8 CAP (Z)	9 TRANS (DX)	10 TRANS (DY)	11 TRANS (DZ)
8	1	10	3.60000E+02	2	0.00000E-01	0.00000E-01	0.00000E-01	0.00000E-01	0.00000E-01	-8.34465E-07

THIS IS A SWEEP TORUS

POINT 1 (X,Y) : 1.00000E+00 1.00000E+00
 POINT 2 (X,Y) : 1.00000E+00 2.00000E+00
 POINT 3 (X,Y) : 2.00000E+00 2.00000E+00
 POINT 4 (X,Y) : 2.00000E+00 1.00000E+00
 POINT 5 (X,Y) : 1.00000E+00 1.00000E+00

BOOLEANS

1 PART ID	2 OPERATION	3 UNION	4 COLOR	5 COMPONENTS	6 1	7 3	8 1	9 3	10 1	11 3	12 1	13 3	14 1	15 3	16 1	17 3	18 1	19 3
1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19
0.00000E-01	0.00000E-01	0.00000E-01	1.00000E+00	1.00000E+00	1.00000E+00	1.00000E+00	-3.75000E-01	-1.19209E-07	-2.38419E-07									

THIS IS A BOOLEAN

TRUSSES

1 PART ID	2 COLOR	3 LENGTH(X)	4 HEIGHT(Y)	5 WIDTH(Z)	6 SCALEX	7 SCALEY	8 SCALEZ	9 HDIST(DX)	10 HDIST(DY)	11 HDIST(DZ)	12 HDBAY(NHB)	13 HDBAY(NDB)	14 TOP OPEN	15 BOTTOM OPEN	16 LEFT OPEN	17 RIGHT OPEN	18 FRONT OPEN	19 BACK OPEN
10	11	12	13	14	15	16	17	18	19	20	21	22	23	24	25	26	27	28
0.00000E-01	0.00000E-01	5.00000E-01	5.00000E-01	5.00000E-01	2.00000E+00	2.00000E+00	2.00000E+00	2.00000E+00	2.00000E+00	2.00000E+00	0.00000E-01	0.00000E-01	0.00000E-01	0.00000E-01	0.00000E-01	0.00000E-01	0.00000E-01	0.00000E-01

THIS IS A TRUSS

FIGURE 8 (Continued)
EDITOR PRINT FORMAT

4.3.6 COPY

The COPY (C) sublevel command allows the user to create a new part by making a copy of an existing (non-deleted) part. All attributes and orientation parameters of the part to be copied will be duplicated in the new part.

After issuing the COPY command, the user is requested to enter the part identification number of the part to be copied or a QUIT (or Q) to return to the EDITOR command level. If an invalid part number is entered (i.e. the part is deleted or non-existent) then the request is repeated.

A menu of five options is displayed next, and the user must enter an integer between 1 and 5 which corresponds with the desired option:

- 1 TO PRINT PART DESCRIPTION
- 2 TO PRINT PART DESCRIPTION AND DISPLAY PART
- 3 TO CONFIRM COPY
- 4 TO MAKE NEW SELECTION
- 5 TO RETURN TO THE EDITOR

The first two options allow the user to verify that the selected part I.D. number matches the part to be copied by printing the part definition (see section 4.3.5), or by printing the part definition and displaying the part (see section 4.4.2). The third option allows the user to confirm the copy and automatically enter the MODIFY sublevel command

(see section 4.3.2). The fourth option allows the user to select a different part for copying. Finally, the fifth option returns to the EDITOR command level without copying the part.

4.4 DISPLAY COMMAND LEVEL

The DISPLAY (D) command sublevel permits the designer to view selected components of the geometric model throughout the creation and modification processes. Subject to the physical limitations of the display device, the user can designate either wire frame or shaded image renderings of the model.

Embedded within the DISPLAY sublevel is a graphics editor. The SMP GRAPHICS EDITOR provides the designer with a mechanism for transforming primitives and querying the model geometry through the manipulation of an interactive graphical input device.

The complete list of viewing operations follows:

- R - RESET VIEWING OPTION
- M - VIEWING OPTION MENU
- D - DRAW MODEL
- T - DISPLAY FOUR VIEWS OF MODELS
- Z - ZOOM
- S - HIDDEN SURFACE
- C - COMPOSITE HIDDEN SURFACE
- G - GRAPHICS EDITOR

These operations are detailed in subsequent sections.

The general viewing options, set by the "R" and "M" commands, impact the other viewing operations. The viewing options include: setting the global transformation parameters, selecting the form in which "back faces" are to be depicted,

toggling the "shrink" switch, and opting for part labels. The terminology describing these viewing options requires further explanation. The global transformation matrix (rotation and scaling only) determines the spatial orientation for the entire model. (Note that the global transformation is not retained in either the parts-file or geometry-file). Recalling the boundary representation scheme (see section 3.2), a "back face" is a planar section of a primitive whose normal points away from the "observer" (the viewing surface). The "shrinking" operation is the standard finite element method (FEM) technique for separating vertices, edges, and faces. Finally, the user can "mark" each part with its identification number via the part labeling option.

4.4.1 RESET COMMAND

The RESET (R) command serves to initialize or reset the general viewing transformation and options.

Employing the terminology of the preceding section, the global transformation matrix is set to an identity matrix. No distinction is made between "front" and "back" faces. Element (edge) "shrinking" is disabled, and parts are not marked with their identification numbers.

RESET does not preselect parts for viewing; this responsibility lies with the module that created the part (either the EDIT commands (see section 4.3) or the READ command (see section 4.1)) or with the MENU command discussed in the next subsection. However, RESET does deselect all component parts.

4.4.2 MENU COMMAND

The MENU (M) command provides facilities for: designating which parts of the model are to be displayed, modifying the global transformation matrix, and specifying the desired viewing options.

The system prompt

DESIGNATE THE "PARTS" TO DISPLAY

signals the user to select parts for viewing. As discussed previously (see section 4), an "*" (implying all parts) or a list of parts separated by a "," (individual parts) and/or "-" (part ranges) is input to select parts.

The user is next prompted to

ENTER "Y" TO CHANGE THE VIEWING TRANSFORMATION.

An affirmative response results in the user being prompted to

ENTER ROTATION ANGLES IN X,Y,Z

and then

ENTER SCALE FACTORS IN X,Y,Z.

The transformations are applied in the following order: X rotation, Y rotation, Z rotation, X scale, Y scale, and Z scale. The resulting transformation matrix remains the current global transformation until either the RESET (see section 4.4.1) or MENU commands are reissued. Recall that

transformation values are not accumulated on successive calls to MENU. Because there is no "clipping" of the model, with the exception of the ZOOM command (see section 4.4.5), there is no provision for global translation.

The system prompt

ENTER "Y" TO CHANGE VIEWING OPTIONS

signals the user that options controlling the display of "back faces", printing of part labels, and "face shrinking" can now be modified. The removal (cull) of "back faces" can simplify a wire frame display of the model. SMP provides the user with three choices for accommodating the display of "back faces" prompted as follows:

```
ENTER BACK FACE CULL OPTION
-1  NO CULL
0   HIDE BACK FACES
1   DASH BACK FACES
```

The prompt

ENTER "Y" FOR PART LABELS
OR "N" FOR NO PART LABELS

identifies the "part label" option. A "Y" input results in the part identification number (see section 3.2) being output at the part "centroid" (see section 2). The "shrink" option is used to separate the individual faces within a part by shortening the edges of the face around its centroid. An affirmative response to the prompt

ENTER "Y" FOR ELEMENT SHRINKING

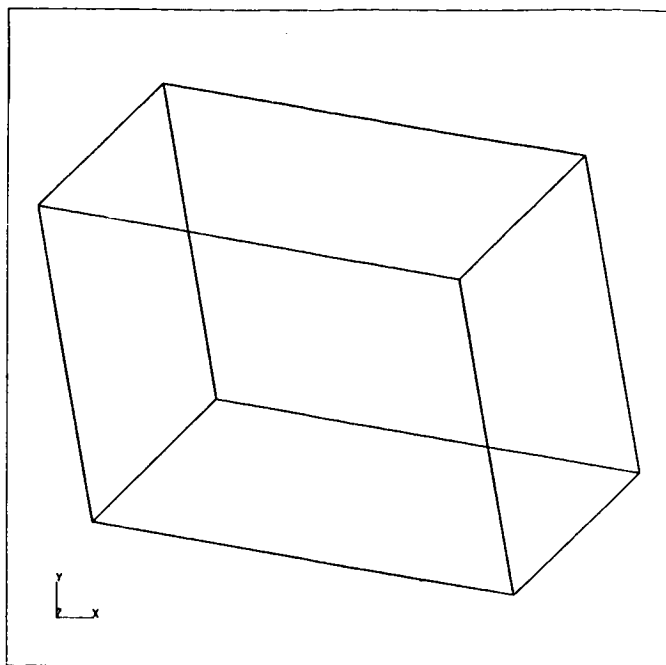
OR "N" FOR NO SHRINKING

requires the user to

ENTER SHRINKAGE FACTOR [0,1]

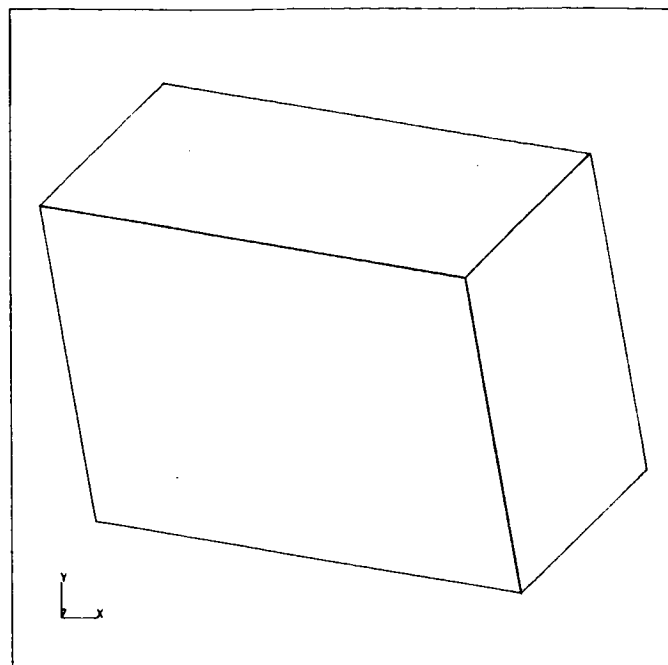
Increasing the "shrinkage factor" increases the extent of the shrinking, from no shrinking (0) to total shrinking (1).

The "back face cull" and "shrink" options are illustrated in FIGURE 9. FIGURE 9(a) shows a primitive "box" with no cull. FIGURE 9(b) shows the same "box" with a cull, and FIGURE 9(c) depicts the "back faces" by dashed lines. The "shrink" option is illustrated in FIGURE 9(d).



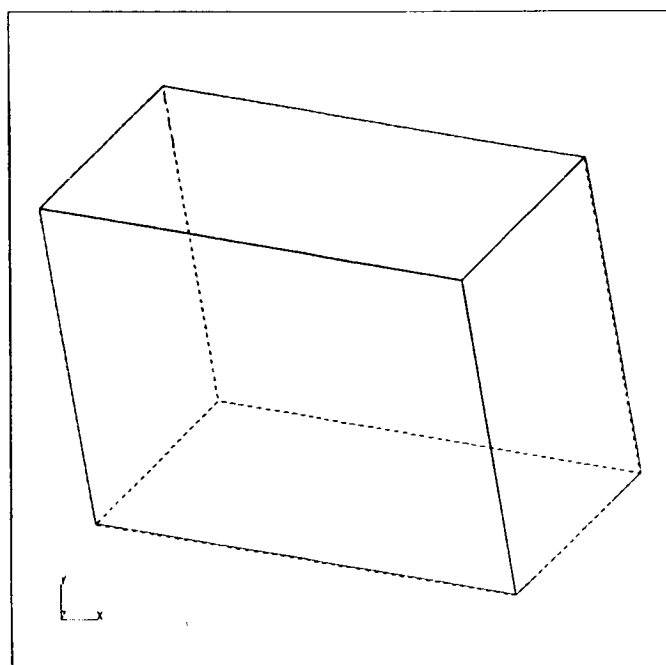
(A)

NO BACK FACE CULL



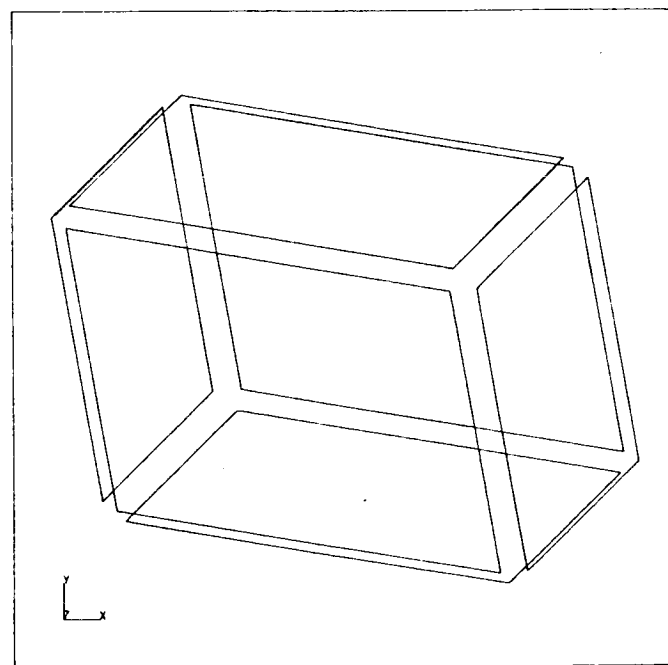
(B)

BACK FACE CULL



(C)

DASH BACK FACES



(D)

SHRINK FACES

FIGURE 9

4.4.3 DRAW COMMAND

The DRAW (D) command generates a "wire-frame" drawing of the geometric model subject to the part selection, global transformation, and viewing options specified by either the RESET (see section 4.4.1) or MENU (see section 4.4.2) commands.

The DRAW command does not have any options. If the display device supports color, the parts are rendered in their assigned colors.

The 3-D axes positioned in the lower left corner of the display provides the user with the current global orientation. (Recall that SMP utilizes a right-handed coordinate system with the default view being the xy-plane and positive z-axis facing the viewer.)

If the "wire-frame" rendering appears too cluttered, the "back face cull" option (see section 4.4.2) may simplify the image. If specific sections of the model are too small to resolve, selecting specific parts (see section 4.4) and viewing only a partial model may be appropriate.

4.4.4 FOUR VIEW COMMAND

The FOUR VIEW (T) command expands the DRAW command (see section 4.4.3) by automatically providing four distinct views of the geometry model subject to same specifications as DRAW.

The display surface is subdivided into four equally sized regions. The default view (xy-plane) is displayed in the upper left region. A yz-plane view is displayed in the upper right region, and a zx-plane view is displayed in the lower left region. The lower right region views the model from the current global orientation as specified in the MENU command. If MENU has not been called, a default rotation of X,Y,Z=45 degrees is supplied.

The FOUR VIEW command is not subject to options.

FIGURE 10 illustrates the FOUR VIEW command.

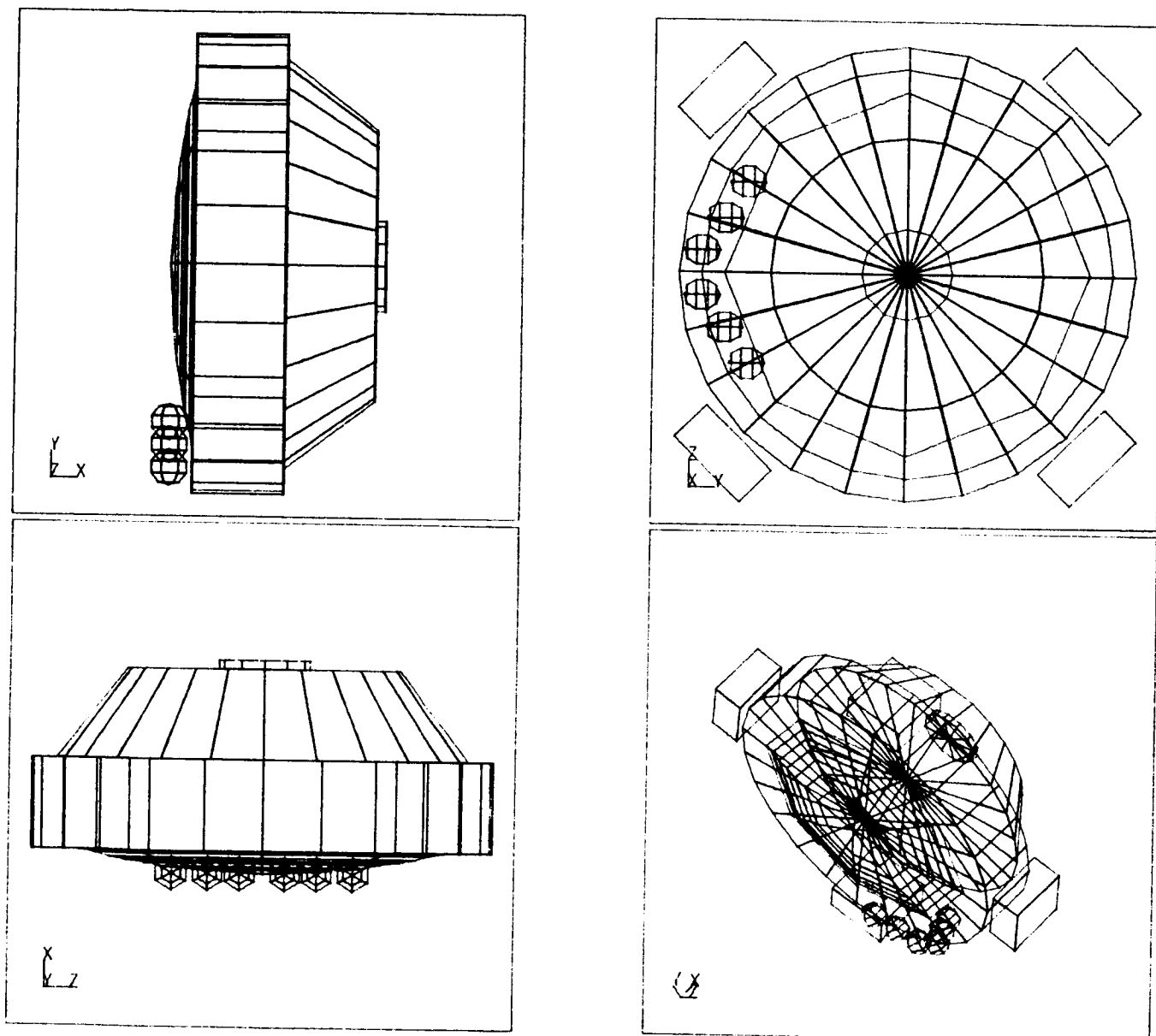


FIGURE 10
SPACE STATION FOUR VIEW

4.4.5 ZOOM COMMAND

The ZOOM (Z) command permits the designer to "zoom-in" on a specified region of the geometric model. The command requires interactive user input implying that the command is applicable only to those graphics terminals which support one or more graphical input devices.

The user is initially prompted as to whether the model should be redrawn. (A negative response is a time-saver if the image is already viewable.) The redrawn view is oriented by the current global transformation matrix and is subject to current part selections and viewing options.

SMP prompts the user to

PICK REGION LL CORNER

and

PICK REGION UR CORNER

which correspond to the lower left ("LL") and upper right ("UR") corners of the rectangular region of the model to be zoomed. The specified zoom region is depicted by a dash-lined rectangle, and the user is required to:

ENTER Y (ACCEPT)
OR N (REJECT)
OR R (RETURN).

A "Y" response results in the display of the zoomed region of

the model. If the region is unacceptable, a "N" response forces a return to the zoom region selection process. A response of "R", causes an exit to the DISPLAY command level.

Because the default "window" is not reset upon exit from ZOOM, the zooming process can be repeated to view a still smaller region of the model via repeated calls to ZOOM. References to either RESET (see section 4.4.1) or MENU (see section 4.4.2) cause the default "window" to be restored.

4.4.6 HIDDEN SURFACE COMMAND

The HIDDEN SURFACE (S) command renders the geometric model as a shaded image with all hidden surfaces removed. This command is intended for use on raster graphics devices (either color or monochrome) where some type of "imaging" is possible. However, a hidden line option is available for vector storage tubes or refresh devices.

The hidden surface calculation is an adaptation of the "priority" algorithm of Newell [11]. The user can invoke either of two shading models: "flat" or "smooth". The "flat" (or "constant") shading model is based on the model described by Foley [12], and the "smooth" shading model is a variant of Gouraud's work [13].

The user is initially prompted to enter the display option:

```
ENTER  S - ELIMINATE HIDDEN SURFACE
        L - ELIMINATE HIDDEN LINES
        B - BOTH (SEPARATE VIEWS)
        R - RETURN.
```

A response of "S", "L", or "R" results in an action suggested by the prompt. (However, a selection of "S" on a non-raster device is automatically changed to an "L" without any informative messages.) If both a hidden surface and hidden line rendering are desired, the "B" response should be selected. In this case, two views are produced without user

intervention: the hidden line view followed by the hidden surface view.

Limitations of the available graphics display hardware resulted in the inclusion of special purpose image display options. These options relate to: dithering, saving the image on a disk file, changing the computation resolution, and opting for an alternate color table. The user is prompted to

ENTER "Y" TO OVERRIDE DEFAULT
IMAGE DISPLAY OPTIONS.

Notice that there are no "current" values for these options; any negative response always results in the default values being used.

An affirmative response results in the user being prompted to

ENTER "Y" TO APPLY DITHER FOR SHADING

Entering "N" invokes the default action. If a "Y" is specified, the image is displayed using an ordered dither technique. On devices which support less than 256 colors, this is the only way to produce a satisfactory hidden surface rendering from SMP.

Next, the user is prompted to

ENTER DISPLAY/DISK IMAGE OPTION
0 - IMAGE OUTPUT TO DISPLAY
1 - IMAGE STORED ON FILE "SMPIMAGE"

Entering "0" invokes the default action. If "1" is specified, the image is written to the file SMPIMAGE. This file can be later post-processed via a local MOVIE.BYU [9] post-processor program. The option is most applicable to generating a shaded image while working from a non-raster display device.

The third option permits the user to change the computation resolution to be other than the default device resolution. This option is normally used in conjunction with the first option when the resolution of the "target" display device differs from the present display device.

The prompt

ENTER RESOLUTION CHANGE OPTION
N - DEFAULT TO DEVICE LIMITS
Y - REDUCE FOR FASTER DISPLAY

"N" is the default response. As the "Y" prompt implies, reducing the computation resolution will reduce the CPU and "wall-clock" time necessary to generate the image. If a "Y" response is received, the user must

ENTER NEW RESOLUTION (SQUARE ONLY)
RESOLUTION FOLLOWED BY STARTING PIXEL

The "square" restriction is intended to prohibit image distortion. The "resolution" (IMAX) and "starting pixel"

(IMIN) input cause the image to be displayed on the screen in a square of dimension (IMAX-IMIN) with lower left corner at (IMIN, IMIN). Every effort is made to detect and correct illegal responses.

The final option relates to overriding the assigned part colors (see section 2) and employing a "grey scale" color table. This option is useful (but not necessary) only if the "smooth" shading option is selected. The prompt

```
ENTER COLOR TABLE OPTION
  0 - USE ASSIGNED PART COLORS
  1 - OVERRIDE WITH GREY SCALE
```

signals the option. The default response is "0".

The shading options and parameters must now be specified. The reader should refer to references [12] and [13] for pertinent background information. The user is initially required to

```
ENTER SHADING OPTION
  0 - FLAT ELEMENT SHADING
  1 - SMOOTH ELEMENT SHADING.
```

Because of the reduced computation time, "flat" shading is usually preferred. If the "flat" shaded image does not produce the desired realism, "smooth" shading should be employed.

After the shading model has been selected, the default shading parameters can be modified. The prompt

ENTER "Y" TO CHANGE SHADING PARAMETERS

triggers this action. An affirmative response permits the user to

ENTER DIFFUSED LIGHT VALUE

and

ENTER REGULAR LIGHT EXPONENT.

The "diffused" (ambient) light coefficient defaults to 0.30 and must reside in the interval [0,1]. The regular light exponent defaults to 1. Since shading models are more empirical than analytical, choosing the proper values for these parameters may require experimentation.

The final option is the "back face cull" (see section 4.4.2) option. The user is prompted to

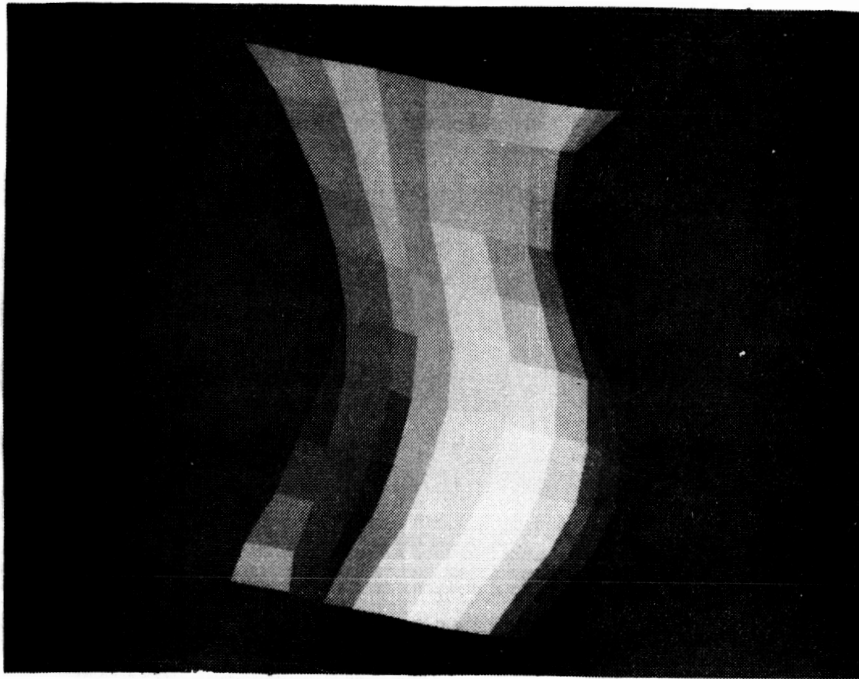
ENTER BACK FACE CULL OPTION
N - NO BACK FACE CULL
Y - PERFORM BACK FACE CULL
R - RETURN.

A "Y" response will normally reduce the number of faces that must be processed; and thus speed-up the display process. However, this option is not always desirable on objects with "holes". Notice that the "R" response affords the last opportunity to "return" to the DISPLAY command sublevel.

The distinction between "flat" and "smooth" shading is illustrated in FIGURE 11. The surface patch is generated external to SMP and later input as an "external" part (see

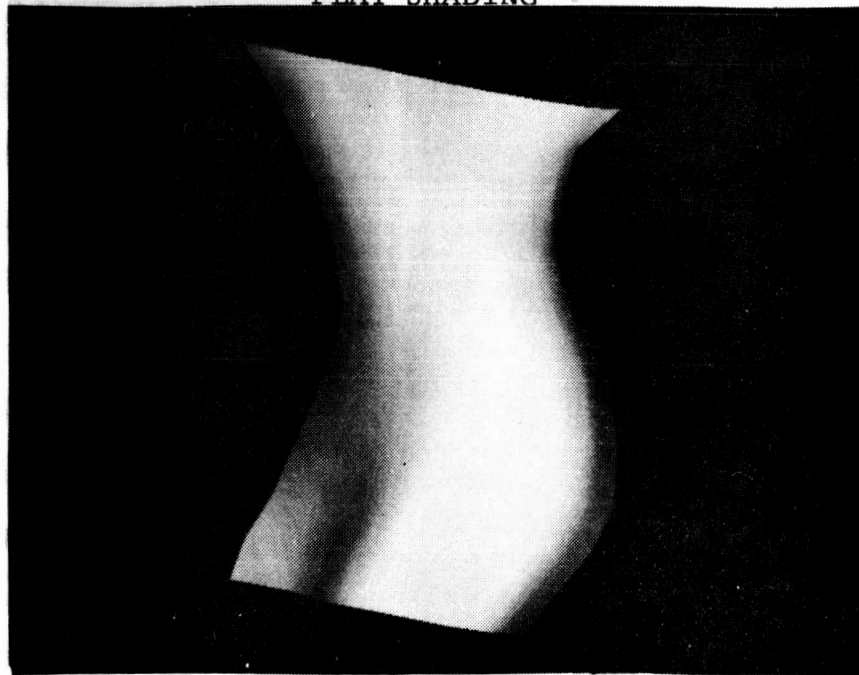
section 2.4). The effect of "flat" shading is shown in 11(a) and the effect of "smooth" shading is depicted in 11(b).

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(A)

FLAT SHADING



(B)

SMOOTH SHADING

FIGURE 11
SUREFACE SHADED IMAGE

4.4.7 COMPOSITE HIDDEN SURFACE

The Composite Hidden Surface (C) command combines the Four View command (see section 4.4.4) with the Hidden Surface command (see section 4.4.6). The result is four hidden surface renderings of the geometric model using the display surface layout and orientation selections discussed in section 4.4.4. The user is required to specify the hidden surface options as described in section 4.4.6. These options remain in effect for all four views.

4.4.8 GRAPHICS EDITOR

A major goal of the SMP system is to free the user from direct interaction with lower level details of the model geometry (such as coordinates of vertices and connectivity of faces) and allow him to manipulate the model through higher level dimension and construction attributes (such as the length, width and height of a box). Continuing this philosophy, the GRAPHICS EDITOR (G) allows manipulation of model parts through direct interaction between the two dimensional (2-D) projection of the model on the screen and a graphics input device [12]. Locations of one or more screen positions (determined by positioning the graphics cursor) are transformed by the system into a three dimensional (3-D) point that may be identified with some point in the "model space" (such as a specific part centroid, vertex, or a new position). The user may modify (alter or orient) the model using visual cues while requiring no knowledge of actual boundary geometry. Should knowledge of point coordinates become important, a secondary function of the GRAPHICS EDITOR provides a mechanism for using the graphics cursor to first locate a point and then return the values of its coordinates.

All models constructed with SMP are three dimensional but must obviously be projected onto a two dimensional display screen.

A graphics input device may return the 2-D coordinates representing the screen position of the graphics cursor. This approach presents a fundamental problem for graphical editing: how may a 2-D location be used to identify a 3-D point?

The GRAPHICS EDITOR resolves the uniqueness problem for the 3-D point by allowing the user to "edit" multiple views of the model. Upon issuing a GRAPHICS EDITOR command, the user is prompted to enter a choice of view (either the XY, YZ, or ZX plane). A display of the model in the selected view is then generated, and the graphics cursor appears. The user positions the cursor to the point to be "picked" and "triggers" the graphics input device. If the desired point is not found ("pick" not close enough to any point), the screen is cleared, and a view in the next coordinate plane is displayed. The user is prompted to try again. On the third failure to obtain a unique point (a failure on all three coordinate planes) a choice is given to abort or to restart the process. Using this method, if two points eligible for picking overlay one another in a particular view, they may be distinguished in a subsequent view (provided they are not coincident). In order to conserve drawing time, when a view is selected, the system checks this choice against the current view. If the views are identical the model is not re-drawn. Also, should the user desire to abort the process, he/she may

do so by entering "Q" when "triggering" the graphics input device.

Currently graphical editing is permitted on only devices : the TEKTRONIX series, and the AED series terminals. In both cases, the graphics cursor is utilized as a "locator" [12] and the "pick" is performed in a manner consistent with the device.

Three basic functions are common to the GRAPHICS EDITOR commands:

- 1.) "pick" a part
- 2.) "pick" a vertex
- 3.) "pick" a new point in space.

All three employ the previously described method for transforming two dimensional screen locations into a three dimensional point.

"Picking" a part requires the user to locate the part "centroid" (which is marked by a "." and the part ID) by selecting a point in close proximity to the part centroid. Should more than one part be returned on the first "pick" (i.e. centroids aligned on an axis perpendicular to the screen), the next view is automatically displayed. However, for the second view, the parts no longer in contention are

displayed with dashed lines (thus simplifying the user's choice). This may be repeated for the third view if a unique part has not been returned.

The part selection procedure is illustrated in FIGURE 12(a) and FIGURE 12(b). The centroids of the "box" (part ID 1) and "sphere" (part ID 2) are coincident in the XY-plane view of FIGURE 12(a). A "pick" near this centroid results in the display of FIGURE 12(b). The "sphere" and "box" remain but the "cylinder" (part ID 3) is "dashed" resulting in a unique choice on the second attempt.

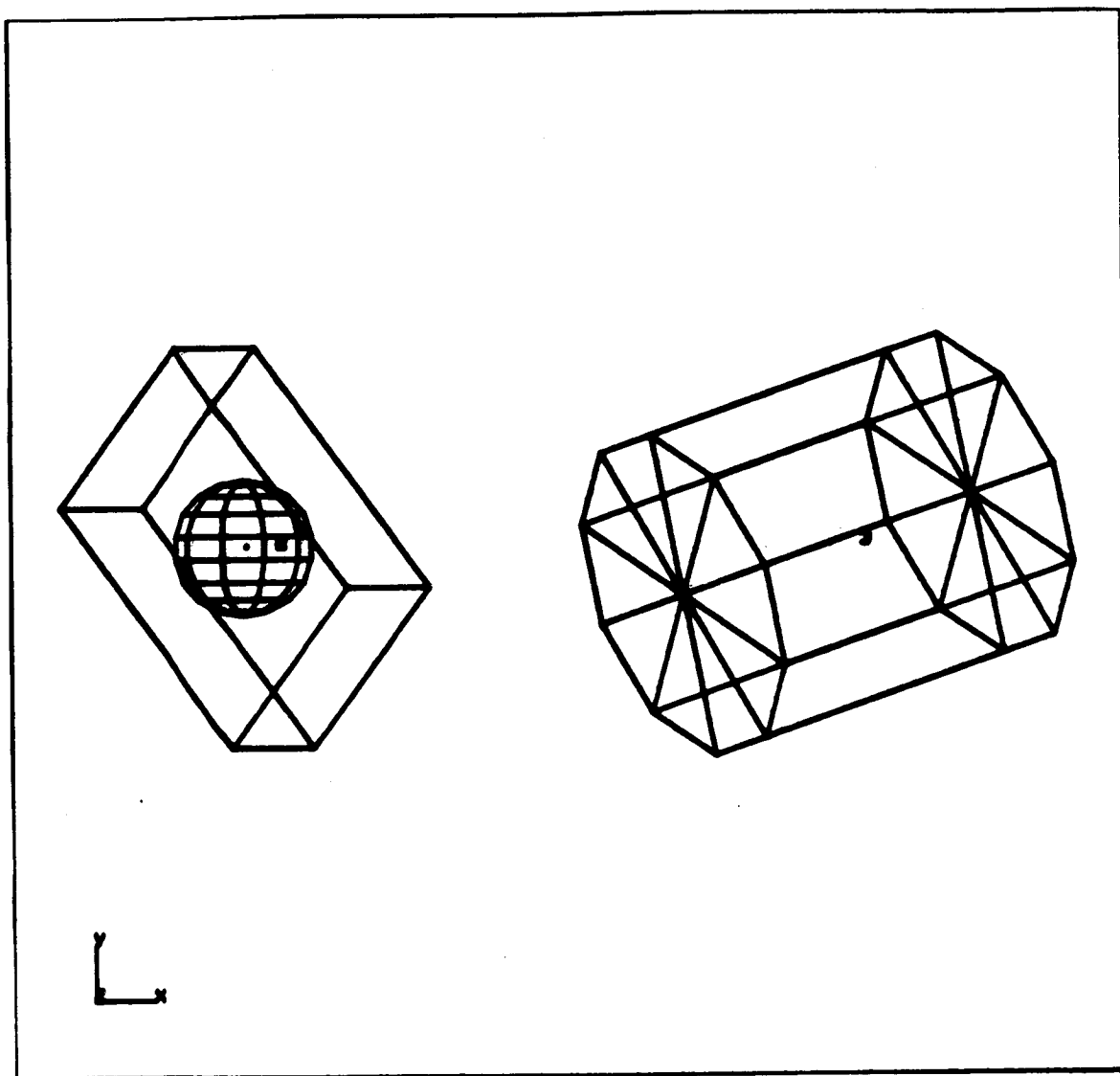


FIGURE 12a
GRAPHICAL PART SELECTION - XY PLANE VIEW

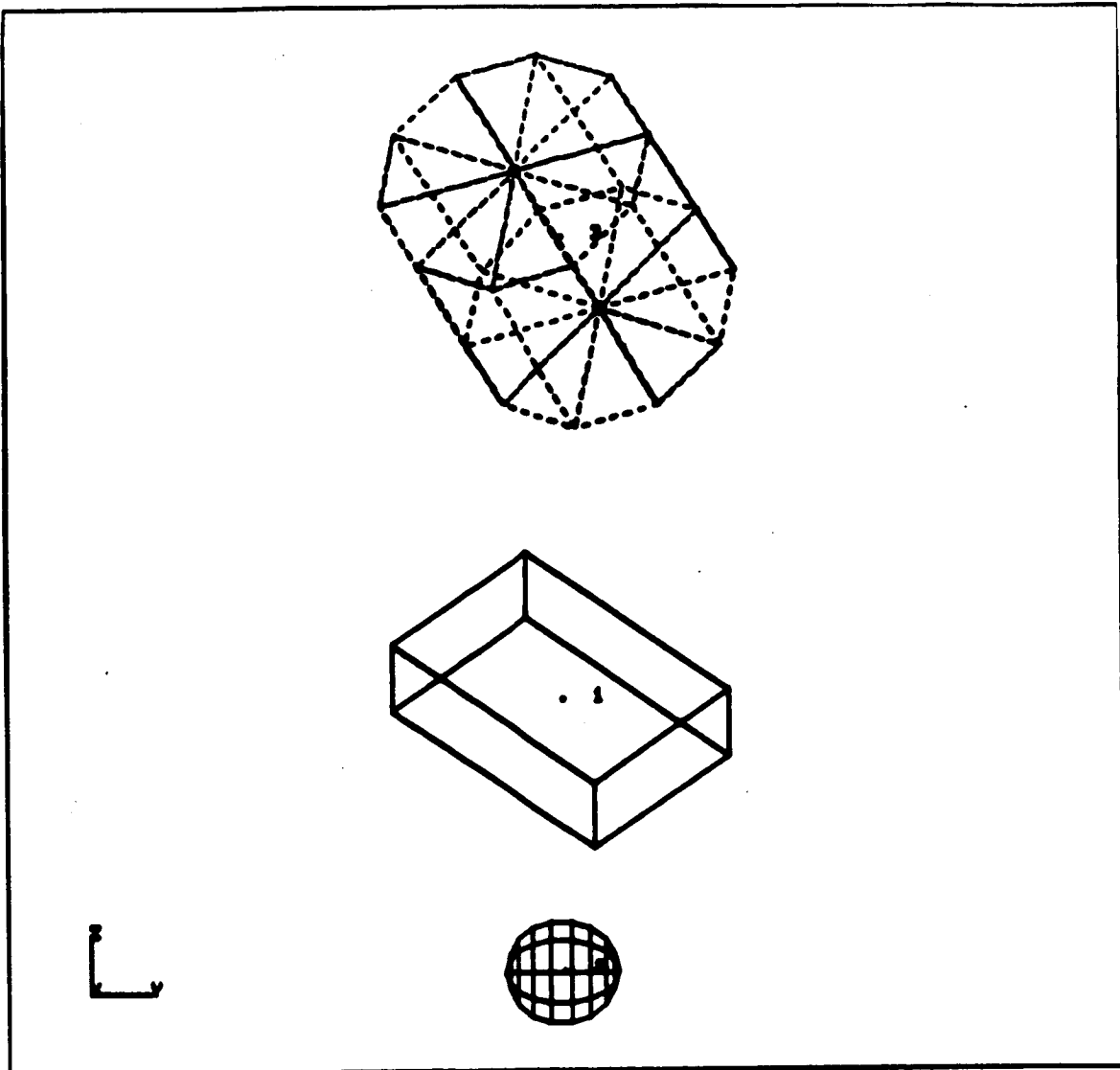


FIGURE 12b
GRAPHICAL PART SELECTION - YZ PLANE VIEW

"Picking" a vertex first requires the user to graphically identify a part. He/she must then locate a three dimensional point "near" the desired part vertex. Similar to identifying a part, "picks" are made in the three coordinate planes until a unique point is identified. Also, after the first pick, the only eligible points are those returned from the previous pick (a "dashed box" is drawn around all eligible points).

Identifying an arbitrary point in model space requires the user to "pick" any desired screen location. The 2-D values returned are assigned to the corresponding coordinates with the third coordinate acquiring the value of zero. The user is then given the opportunity to "pick" from the second view to assign the appropriate value to the third coordinate only.

The following subsections describe the individual GRAPHICS EDITOR commands:

- P - Pick a part
- M - Pick and modify a part
- D - Pick and delete a part
- C - Pick and copy on existing part
- N - Pick and print the coordinates of a point
- T - Translate a part
- R - Rotate a part
- H - Help (documentation)

G - Exit graphics editor

X - Exit program

Each command utilizes the procedure outlined in the preceding paragraphs. The use of the word "identify" in the discussion of the individual GRAPHICS EDITOR commands implies one or more applications of the "location" process.

4.4.8.1 PICK A PART

After "identifying" a part, the part description is printed and the part is displayed.

4.4.8.2 PICK A PART AND EDIT

The EDIT (see section 4.3.2, 4.3.3, and 4.3.6) functions for MODIFYing, DELETing or COPYing a part may be accessed through the GRAPHICS EDITOR by graphically "identifying" the part instead of naming the part by its ID number.

4.4.8.3 PRINT VERTEX COORDINATES

After "identifying" a vertex from an "identified" part, its coordinates are displayed.

4.4.8.4 TRANSLATE A PART

A part may be translated in relation to its centroid or one of its vertices. After "identifying" the desired part centroid or vertex, its new position is "identified" as either an existing vertex or a new 3-D point. The display is subsequently redrawn with the part translated to the new location, and the local transformation matrix is updated.

4.4.8.5 ROTATE A PART

Because rotation is not commutative, this function is currently limited to rotation about the X, Y, and Z axes respectively. For a meaningful rotation to occur, all previous rotations are first eliminated by the system. The user may pick the part and reference point for rotation from any view but when the new point for rotation is picked, the system forces the user to choose first from the YZ view (rotation about the X axis) then the ZX view (rotation about the Y axis) and then the XY view (rotation about the Z axis). After a part and its reference point are picked, a new point is picked which, with the centroid, forms a line. The part is then rotated counterclockwise until the reference point intersects with this line. After each rotation, the user is allowed the options to rotate from the next view, rotate again by the same point in the current view, rotate again by a different point in the current view, display the model in 4 views, or return to the editor.

FIGURE 13(a) depicts three boxes as originally positioned. After selecting a part and a reference vertex for rotation (part #1 and top, right vertex), the user is prompted to supply a new point to form the reference line for rotation (FIGURE 13(b)). The chosen point and line are illustrated.

FIGURE 13(c) demonstrates the positions of the boxes after rotation. Note that part #2 is unchanged from FIGURE 13(a).

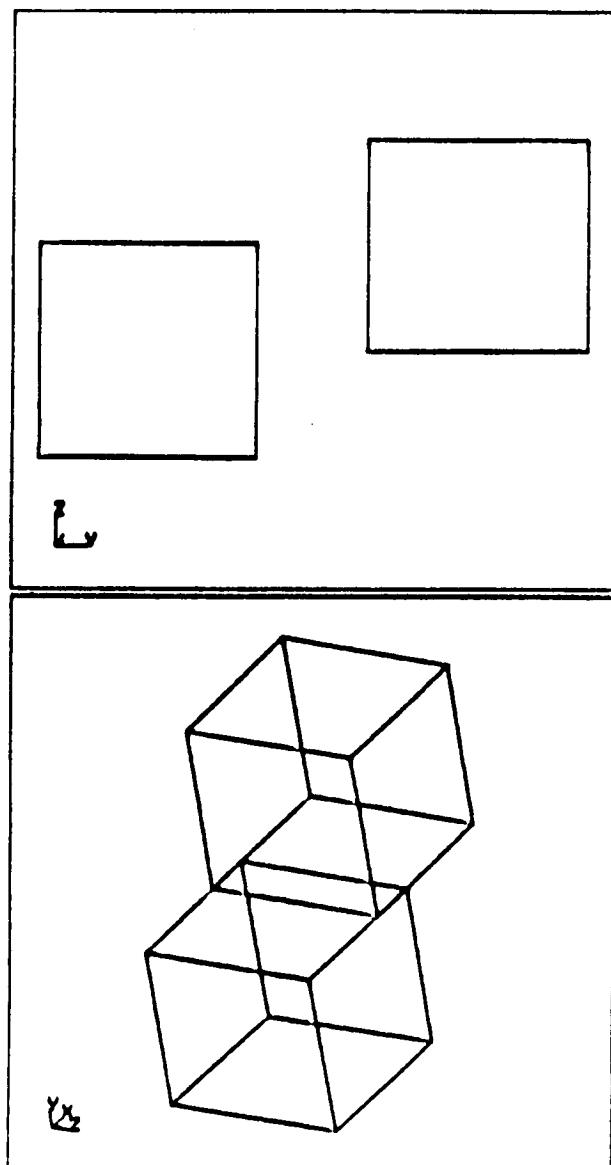
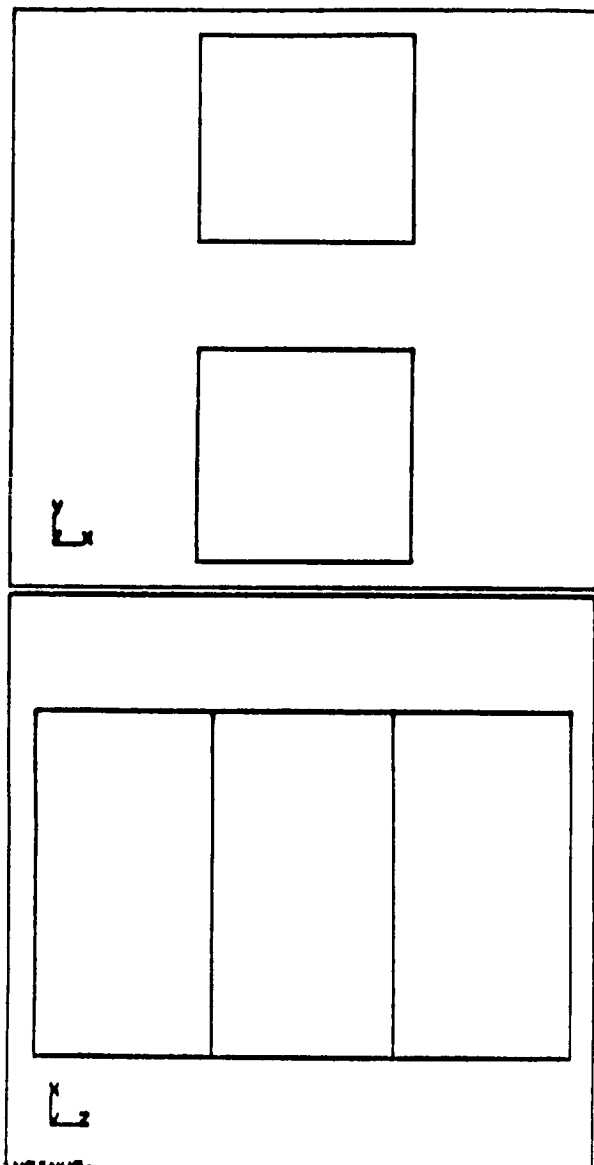


FIGURE 13a
GRAPHICAL PART ROTATION - UNROTATED FOUR VIEW

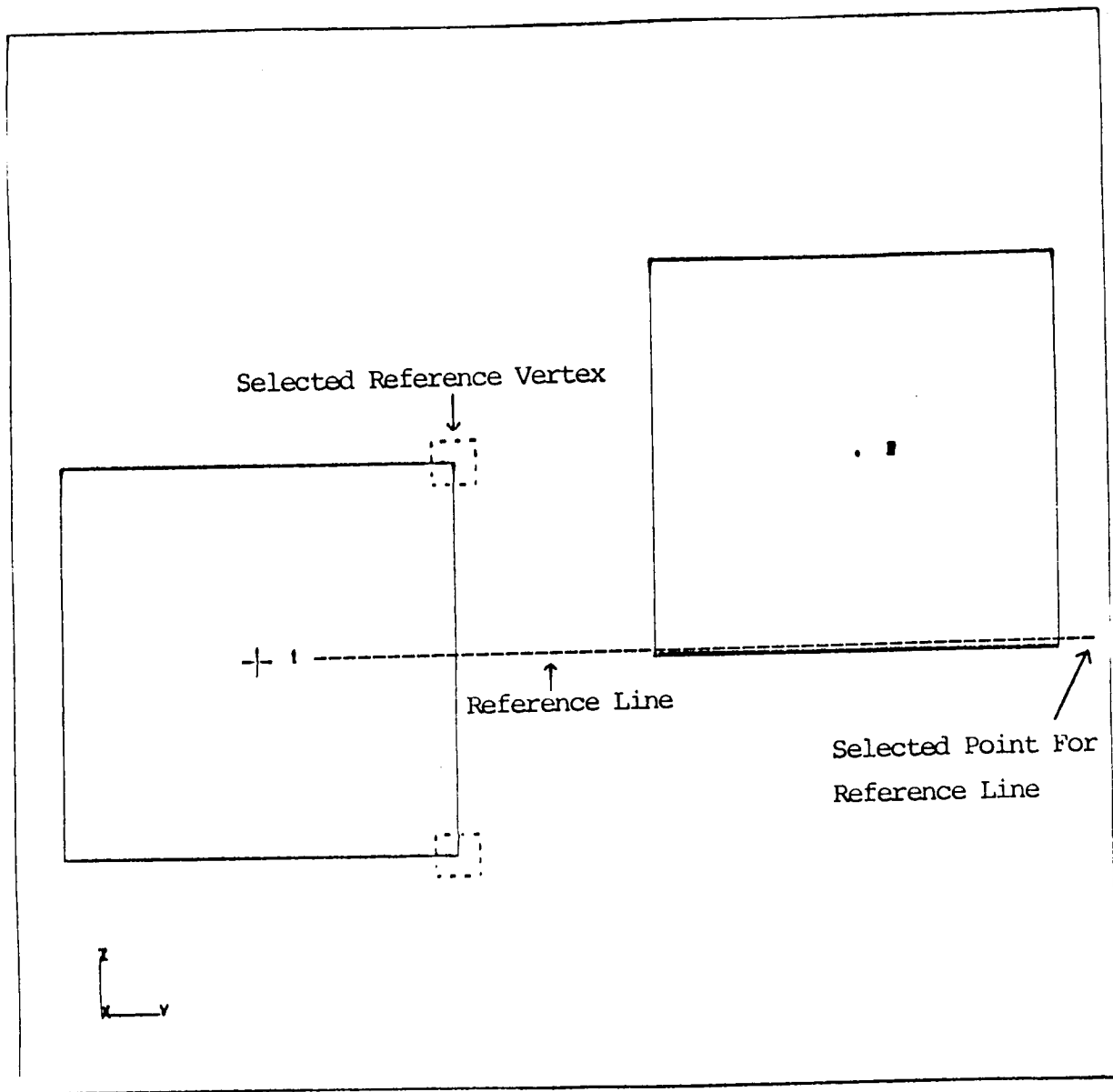


FIGURE 13b
GRAPHICAL PART ROTATION-
REFERENCE VERTEX AND LINE SELECTION

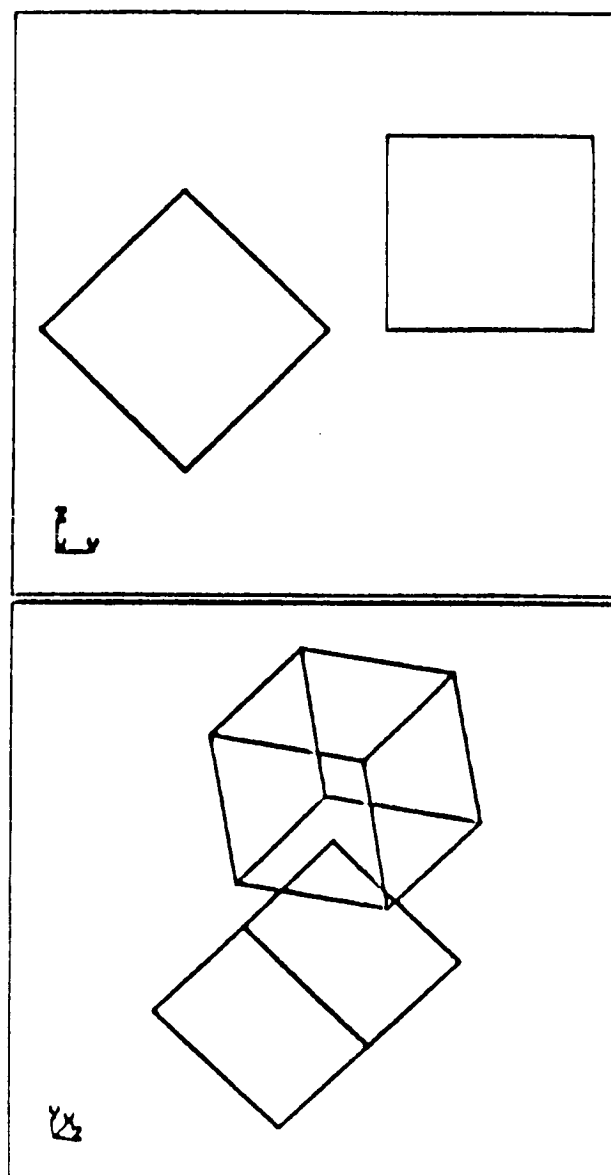
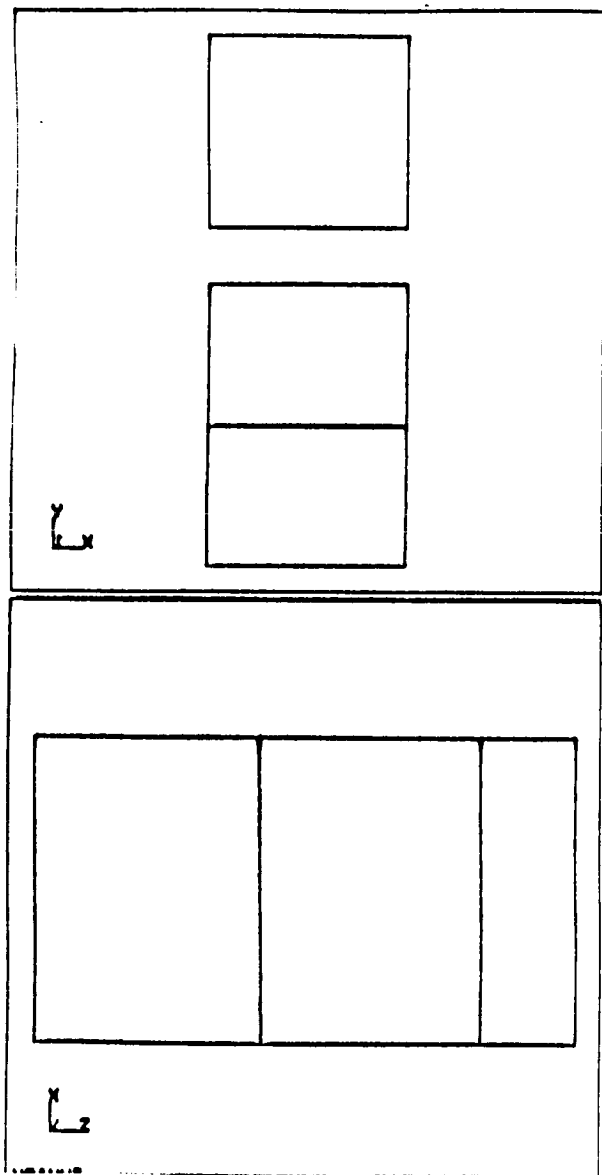


FIGURE 13c
GRAPHICAL PART ROTATION - ROTATED FOUR VIEW

4.5 MODEL ANALYSIS COMMAND LEVEL

A solid modeling package should provide for more than a capability to create, manipulate, and display geometric models. The Model Analysis (A) command serves to fulfill this requirement through the calculations of mass properties and projected areas. The computed mass properties include: mass, volume, moments of inertia, and products of inertia. Although projected areas are not often requested in solid modeling, the application of SMP to spacecraft and space station design necessitated this requirement. The projected area in each of the three coordinate planes is computed.

The computational methods for calculating the mass properties and areas are detailed in the following subsections.

The General Vehicle Synthesizer (GVS) (see section 5.1) provides for an alternate access route to the Model Analysis subcommands and provides an interface to the external world.

4.5.1 MASS PROPERTIES

The Mass Properties (P) command allows for the approximation of volume, mass, moments of inertia, and products of inertia for a designated subset of the geometric model. The computed mass properties are numerical approximations of the triple integral

$$\int_s f(x, y, z) \, dv$$

where s represents the solid, dv is the volume differential, and $f(x, y, z)$ is a real valued function which identifies the physical property. TABLE 5 depicts the relationship between the function f and each of the calculated properties.

TABLE 5

MASS PROPERTY FUNCTIONS

<u>PROPERTY</u>	<u>FUNCTION</u>
Volume	1
Mass	p
Moment of Inertia about x axis	$p(y^2 + z^2)/m$
Moment of Inertia about y axis	$p(x^2 + z^2)/m$
Moment of Inertia about z axis	$p(x^2 + y^2)/m$
xy Product of Inertia	pxy
yz Product of Inertia	pyz
zx Product of Inertia	pzx

p is the supplied density

m is the computed mass

The moments of inertia are computed about one of the coordinate axes. The density defaults to unity and may be modified only through the General Vehicle Synthesizer (GVS) (see section 5.1).

The mass property integrals are approximated using the "column decomposition" algorithm described in Lee [15] and [16]. A three dimensional region bounding the solid model is decomposed into a collection of rectangular parallelipeds ("columns") parallel to the z-axis. The number of columns is determined by an arbitrarily-sized rectangular grid in the xy-plane. A ray tracing algorithm, as described in Roth [18], is employed to determine the volume contribution of each column. The contributions of the individual columns are then summed to approximate the integral.

An advantage of the column decomposition scheme is that it's applicable to any solid representation. Numerical accuracy may be improved by refining the grid that controls the decomposition. However, because ray tracing is such a compute intensive operation, increasing the number of columns, and thus the number of rays to cast, may prove time-prohibitive. More specifically, the number of rays cast for an M by N grid is $M*N$. In addition, the ray tracing calculation is directly proportional to the complexity of the designated solid object.

As described later in this section, SMP provides a compromise between numerical accuracy and compute time by providing some user control over the decomposition.

The Mass Properties command initially prompts the user to:

DESIGNATE THE PARTS FOR MASS PROPERTY CALCULATIONS

The selection process is identical to part selection in other command menus (see section 4).

Next the user is prompted to

ENTER "Y" TO APPLY GLOBAL TRANSFORMATIONS

where global transformations are discussed in section 4.4.2.

If the user responds affirmatively, the model subset is re-oriented before the mass properties are computed.

As alluded to previously, the user may wish to trade-off accuracy for efficiency due to the nature of the ray tracing algorithm. This is accomplished by controlling the computation grid size and thus the number of rays which must be cast. For mass properties, only x and y grid sizes are relevant because rays are cast parallel to the z-axis. By default, SMP bounds the designated parts and determines equivalent x and y grid rays so that no more than 100 rays are cast in the vertical or horizontal direction. Empirical evidence indicates that this default number is more than

sufficient for most objects except in objects such as the "truss" (see section 2.1.6) where more rays may be required. The user may override the default grid sizes in one or more directions, but SMP constrains the selection to sizes that will result in more than 10 and fewer than 1000 grid divisions. As a general rule, decreasing the default grid sizes (which increases the number of grid divisions), improves numerical accuracy but increases compute time.

A sample computation grid size editing session follows:

```

ENTER "Y" TO OVERRIDE DEFAULT COMPUTATION GRID SIZE
THE X,Y, AND Z GRID SIZES ARE:
      0.24999991      0.24999991      0.24999991
CORRESPONDING NUMBER OF X,Y, AND Z GRID DIVISIONS ARE:
      100             80             80
Y
ENTER NEW X,Y, AND Z COMPUTATION GRID SIZES
(NOTE: DECREASING DEFAULT GRID SIZES MAY IMPROVE
      ACCURACY, BUT COMPUTATION TIME INCREASES
      EXPONENTIALLY)
.5,.5,.5

```

For complex objects and/or a fine computation grid, interactive computing of mass properties may not be feasible. In this case, the background mode processing of GVS (see section 5.1) may be exercised.

Sample mass properties output for a cylinder (see section 2.1.2) in its default orientation with radius 10, height 40, and using the default grid sizes (see section 4.5.1) is shown in FIGURE 14.

MASS PROPERTY CALCULATIONS
FOR THE FOLLOWING PARTS:

1

VOLUME = 12005.072

MASS = 12005.072

CENTER OF MASS

X = 0.72026802E-02

Y = 0.83955063E-04

Z = -0.27378787E-05

MOMENTS OF INERTIA

X-AXIS = 573836.88

Y-AXIS = 1886996.0

Z-AXIS = 1887576.5

PRODUCTS OF INERTIA

XY = -88.805450

YZ = -0.16405890E-02

ZX = 0.25131490E-01

FIGURE 14

MASS PROPERTIES OUTPUT

4.5.2 PROJECTED AREAS

The Projected Area (A) command approximates the projected area of the solid object in the yz, xz, and xy coordinate planes and also estimates the centroid for each projected area.

The algorithm employed to calculate the areas is a derivative of the column decomposition scheme discussed in section 4.5.1 and also utilizes a ray tracing technique. In the case of projected areas, the ray tracing scheme must be applied once for each area. The solid is bounded in the desired coordinate plane and this region is subdivided into an M X N rectangular grid corresponding to the number of rays that must be cast. However, the algorithm is simplified over the mass properties calculation because the ray casting process is terminated upon encountering the first valid intersection. The areas of the grid cells where rays intersect the solid are summed to approximate the projected area. As with mass properties, the exact ray position within the grid cell is "randomly" determined.

The user interface to the PROJECTED AREA command is identical to the MASS PROPERTY command (see section 4.5.1). The user designates the parts to be considered in the area calculation, decides whether or not to apply the global transformation, and

refines the decomposition grid based on numerical accuracy requirement and compute-time constraints. The 10 minimum and 1000 maximum delimiting the grid size remain in effect. Because the rays are cast parallel to each of the coordinate axes, the grid sizes in all three directions are relevant.

The output from the PROJECTED AREA command for a cylinder in default orientation with radius 10, height 40, and using the default grid sizes is illustrated in FIGURE 15.

PROJECTED AREA CALCULATIONS
FOR THE FOLLOWING PARTS:

1

PROJECTED AREAS

YZ-PLANE = 299.67963
XZ-PLANE = 799.99902
XY-PLANE = 799.99902
YZ-PLANE (Z, Y) = (0.64942185E-02, 0.33022999E-02)
XZ-PLANE (X, Z) = (-0.16982346E-02, -0.94585430E-05)
XY-PLANE (X, Y) = (-0.16982346E-02, -0.94585430E-06)

FIGURE 15

PROJECTED AREAS OUTPUT

5 MISCELLANEOUS COMMANDS

A solid modeling package should provide interfaces to support non-modeling applications. For example, the user may require relevant analysis and/or drafting information to supplement the model geometry. The mass property and projected area calculations described in the previous section (see section 4.5) offer a partial solution, but their application is limited. This section discusses two efforts aimed at supporting non-modeling endeavors.

The General Vehicle Synthesizers (GVS) is a command which provides an interface between the geometry and mass property and projected area data of SMP with an external spacecraft analysis tool, Articulated Rigid Body Control Dynamics (ARCD) program [18]. GVS may be used independent of ARCD as a means of storing mass property or projected area data for future reference.

A "dimensioning" command available under the MISCELLANEOUS command level is useful in determining the relative positions of model primitives. The DIMENSION command will be detailed in a later subsection.

5.1 GENERAL VEHICLE SYNTHESIZER

At NASA Langley Research Center (LaRC), SMP is used in conjunction with the Interactive Design and Evaluation of Advanced Spacecraft (IDEAS) system to design and synthesize future spacecraft concepts [19]. As IDEAS scope of application expanded from antenna design to space station design, the requirement arose for an interface between the solid geometry of a space station configuration and the software analysis tools of IDEAS. The General Vehicle Synthesizer (GVS) evolved to satisfy this requirement. In particular, the expressed purpose of GVS is to interface to the Articulated Rigid Body Control Dynamics (ARCD) program discussed in Heck [18]. However, the output generated by GVS includes mass property, projected area, and other data that may be utilized in other applications.

The user communicates with GVS (G) through the following command menu:

- D - MODIFY THE DENSITY OF ASSEMBLIES
- P - MASS PROPERTIES
- A - AREAS AND BLOCKAGE
- L - LOCATE ADDED NODES
- I - INITIALIZE GVS SYSTEM
- W - WRITE GVS OUTPUT FILE
- R - READ GVS COMMON DATA FILE
- B - WRITE GVS BATCH COMMAND FILE
- H - HELP (DOCUMENTATION)
- Q - EXIT GVS SUBLEVEL
- X - EXIT PROGRAM

These commands are explained in the succeeding subsections. Before proceeding with individual command explanations, some preliminary information concerning GVS input and output is required.

GVS parts are designated as articular parts (A/P) or spacecraft parts (S/C). To uniquely identify these parts, the first three characters of the 80 character part description (see section 2) are reserved for the A/P or S/C designation. Parts not identified in this manner are ignored by GVS.

If GVS parts are "composite" parts (assemblies (see section 2.5) or booleans (see section 2.3)), the component parts must not be tagged as GVS parts.

Articular parts are treated in a special manner because each A/P is associated with a point about which the part articulates. To provide the proper association, each A/P part must be an assembly where the first component is an arbitrarily small "box" (see section 2.1.1) whose centroid represents the articulation point. This technique is useful in dealing with assemblies unrelated to GVS (see section 2.5).

A second issue centers around model editing. The compute intensive nature of the mass property and projected area

calculations (see section 4.5) prohibits the recomputation of this information for every model modification. An alternative is to recompute the GVS output data only when a part's geometry is modified or a new part is added. This decision raised certain data integrity issues that were partially solved by again using the part description. In this instance, the fourth through tenth characters of the description are reserved for a user defined flag designating which GVS parts were modified. The user changes this field each time a part is modified. The final 70 characters of a GVS part description are user specified, but must be unique. The user does not change this field. Summarizing, the GVS part description entry has three fields: GVS part identifier, modification flag, and a unique part description.

COLUMN	123	4	1 0	11 12	8 0
	S/C or A/P	modify flag	part description		

For example, a spacecraft part may have the following description when initially created:

COLUMN	123	4	1 0	11 12	8 0
	S/C	11/23	X SOLAR PANEL		

In this case, the modification flag field (columns 4-10) contains a date. After being modified, the same part may have the following description:

			1	11	8
COLUMN	123	4	0	12	0
	S/C	11/26		X SOLAR PANEL	

In this case, the modification flag field has been changed to the date the modification was made.

If parts are to be grouped into an assembly, the individual part description must not begin with 'S/C' or 'A/P'. The 3-part GVS description should be entered for the assembly description. The following example illustrates this.

			1	11	8
COLUMN	123	4	0	12	0
PART 1	JOI	NT-SP1			
PART 2	SOL	AR PANE		L1	
Assembly	A/P	11/03		SOLAR ASSEMBLY 1	

Understanding GVS output requires exposure to the input for the IDEAS ARCD program. However, ignoring the details for the present, some preliminary commands are useful. ARCD, like many of the IDEAS program modules, communicates data through the RIM relational database management system [20]. GVS interfaces with ARCD by writing to the ARCD relations in a RIM database. The relations are described in a later subsection.

GVS also writes its output to an ASCII file called the GVS common-file. The common-file contains essentially the same information as the RIM database, but is primarily used to save GVS output during the model editing phase. The contents of the common-file are illustrated in a later subsection.

5.1.1 MODIFY THE DENSITY OF ASSEMBLIES

The GVS Modify the Density of Assemblies (D) command is normally the second GVS command selected. This command allows the user to modify either the density or the mass of each GVS part. The default density of each part is 1.0000. The user may change this density value or he may specify the mass of the part. The following prompt is displayed:

ENTER 1 TO MODIFY DENSITIES
OR 2 TO MODIFY MASSES

If the MODIFY DENSITIES option is chosen, the following submenu is displayed:

ENTER 1 TO PRINT ID, DESCRIPTION, AND DENSITY
FOR ALL S/C AND A/P ASSEMBLIES
OR 2 TO MODIFY THE DENSITY OF ASSEMBLIES
OR 3 TO RETURN

If the density is specified, ARCD uses this density and the volume it calculates to calculate the mass of the part.

If the MODIFY MASSES option is chosen, the following submenu is displayed:

ENTER 1 TO PRINT ID, DESCRIPTION, AND DENSITY
FOR ALL S/C AND A/P ASSEMBLIES
OR 2 TO MODIFY THE MASS OF ASSEMBLIES
OR 3 TO RETURN

If the mass is specified, this mass is used in ARCD.

5.1.2 GVS MASS PROPERTIES

The GVS Mass Properties (P) command parallels the Model Analysis Mass Properties command described in section 4.5.1 with two notable exceptions.

The part selection process must include only those articulating or spacecraft parts whose description begins with A/P or S/C, respectively. If the selected part violates this restriction, the user is prompted to repeat the selection process. Normally in GVS, only one A/P or S/C part is selected as opposed to a subset of parts as in MODEL ANALYSIS.

The user may opt to perform the mass property calculations in the "background" mode. This may be accomplished by submitting a "batch" job as detailed in section 5.1.8. The user makes the decision to defer processing by answering the prompt:

ENTER "Y" TO DEFER COMPUTATIONS TO BATCH MODE

An affirmative response "tags" this GVS part for later processing.

The output from the GVS mass property calculations is displayed as shown in FIGURE 14 and is also stored in an

internal data structure for later output to a RIM database
(see section 5.1.6) or GVS common-file (see section 5.1.7).

5.1.3 GVS AREAS AND BLOCKAGE

The GVS projected area and blockage calculations are similar to the projected area calculations in MODEL ANALYSIS (see section 4.5.2) but are directed specifically towards the ARCD program identified in section 5. In addition to the three projected areas, three blockage factors in the range [0.,1.] are determined. The blockage factor is a measure of the "solidity" of a projected area.

For example, a "box" or "cone" has a blockage factor of 1. because its area projection is "solid"; while a "truss" has a blockage factor less than 1 because its area projection contains considerable "empty space". The projected areas and blockage factors are utilized in the so-called "drag" equation of ARCD [21]. The product of the projected area and the blockage factor computed in GVS should approximate the projected area computed in MODEL ANALYSIS.

The user interface to the GVS AREAS and BLOCKAGE (A) command parallels the MODEL ANALYSIS PROJECTED AREA command with the two exceptions noted in section 5.1.2. That is, the part selection is limited to GVS designated parts, and the area and blockage calculations may be deferred.

The projected areas and blockage factors are shown in FIGURE 16 for a 1X4X1 truss (see section 2.1.6) where the element size is uniformly 1, the distance between bays is uniformly 10, and the default grid sizes are used. As with GVS mass property data, the GVS projected areas and blockage factors are stored internally for subsequent output to a RIM database (see section 5.1.6) or GVS common-file (see section 5.1.7).

PROJECTED AREA CALCULATION
FOR THE FOLLOWING PARTS
1

PROJECTED AREAS

YZ-PLANE = 539.99963
XZ-PLANE = 143.99991
XY-PLANE = 539.99951

BLOCKAGE FACTORS

YZ-PLANE = 0.25925922
XZ-PLANE = 0.30555552
XY-PLANE = 0.25925922

PROJECTED AREA CENTROIDS

XY-PLANE	(Z, Y) = (-6.0003633	,	22.498909)
XZ-PLANE	(X, Z) = (5.9999495	,	-6.0000792)
XY-PLANE	(X, Y) = (5.9998369	,	22.498909)

FIGURE 16

PROJECTED AREAS AND BLOCKAGE FACTOR OUTPUT

5.1.4 LOCATE ADDED NODES

The LOCATE ADDED NODES (L) Command is used to locate thrusters, propellant tanks, and AMCD's on a particular part or assembly of a model. The user must apply the restrictions of ARCD when locating these nodes [18].

Nodes may be added, deleted, modified, or displayed. The following prompt is issued to the user:

```
ENTER 1 TO ADD A NODE
      OR 2 TO DELETE A NODE
      OR 3 TO MODIFY A NODE
      OR 4 TO DISPLAY A NODE
      OR 5 TO QUIT
```

5.1.4.1 ADD A NODE

Thrusters, propellant tanks, and AMCD's may be added, using the guidelines for ARCD. These nodes may be added by entering the known coordinates of the location, or by locating them graphically.

ENTER 1 TO LOCATE ADDED NODE BY KNOWN COORDINATES
OR 2 TO LOCATE ADDED NODE GRAPHICALLY

When nodes are located by known coordinates, the user receives the following prompt:

ENTER X,Y,Z COORDINATES FOR ADDED NODE
OR "R" TO RETURN WITHOUT ADDING A NODE

If the first option is chosen, the user enters the coordinates (X,Y,Z) for the added node.

Alternatively, nodes may be located graphically. The user has the option of designating which parts are to be displayed, which nodes are to be displayed, and to change viewing transformations and options. (see section 4.4.2 Display MENU command.)

The user is then prompted with

ENTER 1 TO ASSOCIATE THE ADDED NODE WITH AN
EXISTING "S/C" PART
OR 2 TO ADD A NEW MODE

Option 1 is used primarily when adding propellant tanks or AMCD's as these are usually located at the center of mass of

the spacecraft part. Since the center of mass of the S/C part is displayed, it is very easy to locate this point graphically. If the added node will not be located at the S/C center of mass, option 2 should be used. When option 2 is selected, the user receives the following prompt:

```
ENTER 1 TO IDENTIFY AN  
      EXISTING POINT  
OR 2 TO IDENTIFY A  
      NEW POINT  
OR "R" TO RETURN
```

When the user chooses option 1, the added node is located at an existing point on the faceted model. This option allows the user to choose an exact point, thus giving more accuracy to the location of the added nodes. This is especially useful when locating thrusters. If the user chooses option 2, any point may be identified as the location of the added node. In most all cases, the user will want to choose this point using two different view planes (see section 4.4.8).

The user has the option of choosing the view plane.

```
ENTER 1 FOR XY VIEW  
      OR 2 FOR YZ VIEW  
      OR 3 FOR ZX VIEW  
      OR "R" TO RETURN
```

Being familiar with the model, the user can determine from which view the added node should be identified. Sometimes a unique point cannot be identified from only one view plane. If the user chooses a point on the S/C part in one view plane that actually identifies two or more separate points, the

system automatically displays a different view plane to enable the user to isolate the desired point. Before actually locating the added node, the user is requested to pick the center of the desired S/C part. (see section 4.4.8.)

5.1.4.2 DELETE A NODE

ORIGINAL PAGE IS
OF POOR QUALITY

Nodes are deleted by entering the node ID. The user is given the option to print the added node descriptions to verify the node id. (See section 5.1.4.1 ADD A NODE - Known Coordinates for details.)

NOTE: The option to identify added nodes graphically has not been implemented.

5.1.4.3 MODIFY A NODE

Nodes to be modified may be located either by the node ID or graphically. Modifying nodes is very similar to adding nodes. (see section 5.1.4.1 ADD A NODE for details.)

5.1.4.4 DISPLAY ADDED NODES

Nodes are displayed using two different methods. The first option prints the node description, as follows.

ID	ASSOCIATE	PART	TYPE	POSITION(X,Y,Z)
1		3	THRUSTER	0.00000000 0.00000000 0.00000000

The second option displays, the added node graphically. The user has the option of designating which parts are to be displayed, which nodes are to be displayed, and to change the viewing transformations and options. (see section 5.1.4.1 ADD A NODE-Locate graphically for details).

5.1.5 GVS INITIALIZATION

The Initialize GVS System (I) command is normally the first GVS command selected. This command identifies all GVS parts by searching for the A/P or S/C designation in the part description, initializes the internal data structures used to store the computed mass property and projected area and blockage data, and determines the default grid sizes (see section 4.5) to be used in these calculations.

The user is prompted with:

ENTER "Y" TO OVERRIDE UNITS CONVERSION FACTOR
(DEFAULT IS 0.10000E+01 FOR METERS)

ARCD data must be in meters. If dimensions are in anything except meters, the user must answer 'Y' to this prompt and then enter an appropriate conversion factor when prompted. The INITIALIZE command requires no additional user input.

If the parts-file contains more than the current maximum limit of 50 GVS parts, an informative message is printed and all remaining GVS parts are ignored.

5.1.6 WRITE GVS OUTPUT FILE

The WRITE GVS OUTPUT FILE (W) command produces a user-named GVS common-file and optionally writes the ARCD input data to a RIM database.

The GVS common-file contains the mass property and projected area data generated by GVS. The GVS common-file is an ASCII file whose format is given in TABLE 6.

The WRITE GVS OUTPUT FILE (W) command enables users to write a file and save the information generated by GVS.

The user receives the following prompt:

```
ENTER GVS WRITE OPTION
0 - NONE (IGNORE REQUEST)
1 - GVS MASS PROPERTY/PROJECTED AREA FILE
2 - GVS MASS PROPERTY/PROJECTED AREA FILE AND
    ARCD PROGRAM INPUT DATA TO RIM DATA BASE
```

If the user enters a "1" or a "2", the following prompt appears:

```
ENTER GVS COMMON FILE NAME
OR "R" TO RETURN WITHOUT WRITING A FILE
```

If the file name already exists, the following prompt will appear.

A FILE EXISTS BY THIS NAME.
DO YOU WANT TO OVERWRITE IT (Y/N)

The GVS common-file is written and may be read later to restore previously computed GVS output data. The RIM data base is written to the file(s) DBDATA using the standard RIM naming conventions [20].

5.1.7 READ GVS COMMON DATA FILE

The mass property and projected area data generated by GVS is written to a user-named GVS common-file. Although applicable to external applications, the GVS common-file is normally used during the model editing process to reduce redundant mass property and projected area calculations. More specifically, if the geometry or orientation of a GVS part has not changed, it is unnecessary to recompute its mass properties or projected areas and blockages.

The GVS common-file is an ASCII file whose format is given in TABLE 6. Some of the information residing in this file is relevant only to ARCD program (see section 5). No mention of physical units is made in TABLE 6 because the user is responsible for maintaining a consistent set of units.

The READ GVS COMMON DATA FILE (R) command enables users to name a file containing the information necessary to restore previously computed GVS output data. This command attempts to implicitly identify GVS parts which may have been edited and informs the user accordingly. The identification procedure is accomplished by comparing the "modification" field and the "description" field of the parts-file description with the corresponding fields in GVS common-file (see section 5).

The command-file read utility identifies three differences between the parts-file and the common-file including: added parts, deleted parts, and modified parts. If a GVS part is encountered in the parts-file and not in the common-file, the message

THE FOLLOWING PART EXISTS IN MODEL
BUT IS NOT PART OF GVS FILE

is printed, followed by the part ID and part description, implying this is a new GVS part. Added parts are implicitly tagged for recomputation. For GVS parts identified in the common-file and not in the parts-file, the message

THE FOLLOWING PART EXISTED ON GVS-FILE
BUT IS NO LONGER PART OF THE MODEL

is printed, again followed by the part ID and part description, informing the user that this part was recently deleted. Deleted parts are removed from the common-file on the next re-write.

Added and deleted GVS parts are discovered by encountering part descriptions ("description" field only) in the parts-file that do not match any descriptions in the common-file or vice-versa. In the case of modified GVS parts, the "description" fields of the part description match, but the "modification" fields do not. Consequently, the user must be cognizant of the importance of properly editing the part

description to ensure accurate implicit tagging of GVS parts targeted for recomputation. Explicit tagging of GVS parts may be performed in either the MASS PROPERTIES (see section 5.1.2) or AREAS AND BLOCKAGE (see section 5.1.3) commands.

The user is given a choice when a modified GVS part is encountered. The prompt

THE FOLLOWING "GVS" PART MAY HAVE BEEN MODIFIED

followed by the part ID and part description are printed. The user is then questioned to

```
ENTER RE-INITIALIZATION OPTION
  0 - RE-INITIALIZE GVS PART
  1 - RE-INITIALIZE BUT RETAIN MASS/DENSITY
  2 - NO RE-INITIALIZATION USE GVS FILE DATA
```

These choices provide the user with some degree of flexibility regarding what, if any, data should be retained. Option 0 reinitializes, but resets the density back to the default value of 1.00000. Option 1 reinitializes, but retains the density value the user previously entered. Option 2 should be used cautiously - be absolutely sure the part has not been modified!

The common-file affords considerable potential for reducing mass property and projected area calculation time during model editing. However, to exercise this feature properly, the user must conscientiously remember to edit the description along

with the geometry. Failure to perform this task may result in loss of data integrity between model geometry and model analysis.

ORIGINAL PAGE IS
OF POOR QUALITY

TABLE 6

COMMON FILE DESCRIPTION

<u>RECORD</u>	<u>DESCRIPTION</u>	<u>FORMAT</u>
1	Total number of A/P and S/C parts (<50)	I5
2	GVS part ID number	I5
3	GVS part description	A80
4	GVS part type (0 for S/C, 1 for A/P) ARCD grid point number GVS part density, part mass, part volume	2I5, 3(1X, 1PE12.5)
5	Number of x,y, and z grid divisions unit conversion scale factor	3I5, 5X, E12.5
6	(x,y,z) center of mass (with respect to origin) Moments of inertia along x,y, and z axis	6(1X, 1P, E12.5)
7	xy,yz and zx products of inertia yz,xz, and xy-plane projected areas	6(1X, 1P, E12.5)
8	yz,xz, and xy-plane projected area centroids	6(1X, 1P, E12.5)
9	yz,xz, and xy-plane blockage factors	6(1X, 1P, E12.5)
	(x,y,z) articulation points for A/P parts	6(1X, 1P, E12.5)

(Note: records 2-9 are repeated for each GVS part)

$$N = (\# \text{ of GVS parts}) * 8 + 2$$

N	Number of added nodes	I5
N+1	Added node ID (if associated with assembly) or 0 Added node type (1 thruster, 2 propellant tank, 3 AMCD)	2I5
N+2	(x,y,z) added node position	3(1X, 1P, E12.5)

(Note: records (N+1) and (N+2) are repeated for
each added node)

5.1.8 WRITE GVS BATCH COMMAND FILE

As the name implies, the WRITE GVS BATCH COMMAND FILE (B) command produces a file of SMP commands that enable the mass property and projected area and blockage factor computations of GVS to be performed in the "background" mode. These calculations may be deferred explicitly (see sections 5.1.2 and 5.1.3) or implicitly by modifying existing GVS parts or adding new ones (see section 5.1.7).

In an effort to inform the user of how the "batch" processing is to proceed, certain information is displayed on the user's terminal. First, if the user has positioned propellant tanks (see section 5.1.4) the user is prompted for the propellant tank sizing parameter (KALKTK). This parameter is meaningful only when exercising the ARCD program discussed earlier (see section 5) and will not be discussed in this document (see Leondis [21] for further details).

Next the user is supplied with general and specific information about the nature of the "batch" processing. FIGURE 17 provides a sample of this output. The reader should take note of four important items identified by this command:

- (1) the name of the SMP parts file

- (2) the name of the GVS common-file
- (3) the name of the GVS batch-file and the system command for "submitting" this file
- (4) the name of the file containing the normal GVS output which serves as an audit-trail

GVS does not automatically "submit" the batch job because it could be in error regarding one of the four items listed above. It is the responsibility of the user to verify these four items and then "submit" the batch job upon exit from SMP. If there is an error in one of these four items the user must correct the GVS batch-file using the system editor.

The user is requested to confirm each GVS part for which the mass properties and projected areas and blockage factors are to be recomputed. This confirmation ensures that GVS and the user are in agreement on which GVS parts should be recomputed. The GVS prompts are as follows:

GVS PARTS TARGETED FOR MASS PROPERTY AND PROJECTED AREA CALCULATION IN THE "BATCH" ENVIRONMENT MUST BE CONFIRMED.

WHEN REQUESTED ENTER "Y" TO CONFIRM (ANY OTHER CHARACTER WILL IGNORE THE PART).

ENTER "Y" TO CONFIRM PART NUMBER _____

LEGAL COMMANDS ARE: D - MODIFY THE DENSITY OF ASSEMBLIES
P - MASS PROPERTIES
A - AREAS AND BLOCKAGE
L - LOCATE ADDED NODES
I - INITIALIZE GVS SYSTEM
W - WRITE GVS OUTPUT FILE
R - READ GVS COMMON DATA FILE
B - WRITE GVS BATCH COMMAND FILE
H - HELP (DOCUMENTATION)
Q - EXIT GVS SUBLEVEL
X - EXIT PROGRAM

B

ENTER GVS COMMON FILE NAME
OP "R" TO RETURN WITHOUT WRITING A FILE
TM.COM

ENTER GVS BATCH FILE NAME
OR "R" TO RETURN WITHOUT WRITING A FILE
TM.BAT

THE GVS "BATCH" COMMAND CONSTRUCTS A FILE WHICH
ALLOWS SMP TO BE EXECUTED IN THE "BACKGROUND"
FOR THE PURPOSE OF COMPUTING MASS PROPERTIES
AND PROJECTED AREAS/BLOCKAGES IN GVS FOR ARCD
PRIOR TO ISSUING THE GVS "BATCH" COMMAND, THE
FOLLOWING ACTIONS SHOULD BE COMPLETED:

- . ALL "ADDED" NODES SHOULD BE ADDED,
- . ALL RELEVANT ASSEMBLIES SHOULD BE
DESIGNATED AND WITH MASS/DENSITY OPTION
AND COMPUTATION GRID SIZE SPECIFIED

BASED ON PREVIOUSLY SUPPLIED INFORMATION, THE
FOLLOWING ASSUMPTIONS ARE MADE:

- . PARTS-FILE NAME = TM.PARTS
- . GVS COMMON-FILE NAME = TM.COM
- . THE "IDEAS" DATA BASE RESIDES IN THE USERS
CURRENT WORKING AREA

AFTER EXITING (INTERACTIVE) SMP, (BACKGROUND)
SMP IS EXECUTED BY ENTERING:
SUBMIT/KEEP TM.BAT

UPON COMPLETION OF (BACKGROUND) SMP, THE "IDEAS"
DATA BASE IS UPDATED
AN AUDIT TRAIL OF THE "JOB" MAY BE REVIEWED BY
LISTING THE FILE:
GVSLOG.LOG

FIGURE 17

BATCH PROCESSING

5.2 DIMENSIONS BETWEEN PARTS

Using functions of the GRAPHICS EDITOR (see section 4.4.8) the user may request the distance between two part "centroids" or between two selected vertices (on the same or different parts). In addition to the absolute distance between the selected points, the X, Y, and Z displacements are also supplied.

The user may also specify a "range of parts" (see section 4). The part labels of the designated parts are displayed and line segments connecting these part centroids are drawn. The length of each line segment is then displayed at its midpoint.

6 SOFTWARE AND USER ENVIRONMENTS

The preceding sections present the functionality of the SMP software. The intent of this section is to describe the modeller from a utility perspective by discussing the software and user environments.

No attempt is made to describe the software algorithms or to teach the "host" operating systems. Instead, this section identifies the computing environments surrounding the modeller and assists the designer with the use of SMP in such an environment.

6.1 SOFTWARE ENVIRONMENT

The SMP package was originally developed on a PRIME 850 minicomputer and recently implemented on a VAX 11/785 minicomputer. It relies heavily on the virtual memory facilities available within both the PRIMOS (Revision 19.2) and the VMS (Version 4.2) operating systems. The host interfaces are discussed in section 6.1.2.

The modeller is designed to be exercised in an interactive environment with strong dependence on graphics for user feedback. SMP is interfaced to the following graphic display devices:

- ADAGE 3000 Color Raster Display System

- Advanced Electronics Design (AED) 512/767/1280

- Intelligent Systems Corporation (ISC) 8001R

- Tektronix 4100 Series

- Tektronix 4010 Series

Additionally, an interface exists for a non-graphics, alphanumeric terminal. The device interfaces are described in section 6.1.3.

6.1.1 SOURCE-LEVEL LANGUAGE

The software is coded exclusively in FORTRAN 77 with relatively few machine dependencies. These non-ANSI exceptions evolved for reasons of efficiency and are clearly documented as comments in the source code. One such example is the use of the FORTRAN intrinsic bit manipulation functions. Another is the use of host dependent FORTRAN-callable subroutines to accomplish input/ output tasks more efficiently. These host dependencies are discussed further in the next section.

The modeller source code is organized functionally into thirteen separate modules. For example, the graphics device dependent code is localized for the most part in one module to facilitate implementation of new devices. In addition, the modeller is supported by several utility and device level libraries. These are described in Appendix A.

6.1.2 HOST INTERFACES

As noted previously, SMP has been implemented on both the PRIME 850 and VAX 11/785 minicomputers, and relies heavily on the virtual memory feature of both the PRIMOS and VMS operating systems. Host dependencies are minimized to facilitate program portability. Those dependencies which do exist are outlined here.

6.1.2.1 PRIME/PRIMOS HOST DEPENDENCIES

The PRIME version of SMP relies on the PRIMOS file management subroutines to perform input/output tasks involving large requests such as reading/writing model geometry. Data transfer via the file management subroutines is generally faster than FORTRAN data transfer. Host independent ANSI-standard FORTRAN I/O requests parallel the PRIME dependent subroutine calls and are clearly marked in each module.

The SMP "interrupt" feature allows the user to prematurely terminate an SMP process and regain program control gracefully. Implementation and use of the "interrupt" feature is host dependent. On PRIMOS, the interrupt is performed by depressing the BREAK key (or <CONTROL> P). When the "interrupt" is issued, the screen is erased and the current menu level prompt is issued. The "interrupt" feature is programmed in the PRIME version of SMP via the PRIMOS condition handling subroutines described in [22].

The GVS BATCH command constructs a command file for executing the GVS mass property and area/blockage computations in a "batch" environment. The command file is opened with a user-supplied filename and contains PRIMOS commands. Upon

completion of SMP, the user submits the command file for processing as a "phantom" process as described in [14].

6.1.2.2 VAX/VMS HOST DEPENDENCIES

The VAX implementation of the "interrupt" feature enables trapping of a '<CONTROL> C' from the user's terminal. It is implemented on a select basis and is only activated while either a hidden surface display or line drawing is being rendered or while mass properties are being computed. A '<CONTROL> C' entered at any other time from the user's terminal is processed in the usual fashion resulting in program termination.

As noted earlier, the GVS BATCH command constructs a command file which allows SMP to be executed in the background for the purpose of computing mass properties. This command file contains VMS commands and, upon completion of SMP, may be submitted for processing as a batch job with the VMS SUBMIT command as described in [24].

The ADAGE 3000 Color Raster Display System is available only in the VAX implementation of SMP and functions in direct memory access mode. The Intelligent Systems Corporation (ISC) B001R interface is not available.

6.1.3 DEVICE INTERFACES

SMP support for display devices is summarized in TABLE 7 and in the following subsections. The graphics device level code is localized for the most part in one module to facilitate implementation of new devices. The Adage 3000 device interface is the only implementation which does not conform to this convention. For each of the display devices, SMP utilizes a device dependent FORTRAN-callable subroutine library to perform device level functions such as MOVE's and DRAW's. In the case of the Tektronix 4010 interface, for example, this device level library is the Tektronix PLOT-10 subroutine package. In most other cases, the subroutine libraries are locally developed modules. Each interface is described in the following subsections.

TABLE 7

SMP DEVICE SUPPORT

<u>Display Device</u>	<u>Screen Space Addressed</u>	<u>Raster Space Addressed</u>	<u>Host Support</u>
Alphanumeric terminal	NA	NA	VAX, PRIME
Adage 3000	x: 0-511 y: 0-511	512 pixels 512 scanlines	VAX
AED 512	x: 0-511 y: 0-511	512 pixels 512 scanlines	VAX, PRIME
AED 767	x: 96-670 y: 0-574	575 pixels 575 scanlines	VAX, PRIME
AED 1280	x: 256-1279 y: 0-1023	1024 pixels 1024 scanlines	VAX
ISC 8001R	x: 96-479 y: 0-383	96 pixels 96 scanlines	PRIME
TEK 4014/4015	x: 243-1023 y: 0-780	NA	VAX, PRIME
TEK 4107/4109	x: 1024-4095 y: 0-3071	480 pixels 480 scanlines	VAX, PRIME
TEK 4115/4129	x: 968-4095 y: 0-3127	1024 pixels 1024 scanlines	VAX, PRIME

6.1.3.1 ALPHANUMERIC TERMINAL

The alphanumeric terminal interface allows the user to perform modeller functions such as reading/writing files, editing, and mass property calculations from any graphics/ non-graphics terminal. The interface is accomplished entirely through FORTRAN I/O and requests for graphical input or output are ignored. The alphanumeric terminal interface is selectable as item 0 from the display device menu and is available in both the PRIME and VAX implementations.

6.1.3.2 ADAGE 3000 COLOR RASTER DISPLAY SYSTEM

The Adage 3000 System is a sophisticated display device equipped with special purpose processors for performing graphics functions such as vector drawing and shaded image rendering in hardware. This local intelligence dramatically improves performance of the graphics dependent functions of SMP. The Adage 3000 was interfaced to SMP by Research Triangle Institute (RTI) under NASA contract. The unique features of the Adage 3000 System as they apply to SMP, including revised user inputs, are detailed in [23].

Because the Adage software is proprietary and the RTI software is totally device specific, this interface is not considered part of the standard SMP package. Instead, a "dummy" library is provided to ensure proper linking of all SMP modules (see Appendix A).

6.1.3.3 ADVANCED ELECTRONICS DESIGN (AED) 512/767/1280

The AED 512/767/1280 model terminals are color raster terminals used in a serial, RS-232C environment. The AED device-level subroutine library was developed specifically for LaRC by Computer Sciences Corporation.

Each model supports 256 colors, so an acceptable hidden surface rendering may be obtained. The screen/raster space limits vary with the model and are shown in TABLE 7. Interactive graphics input is supported via the joystick.

The 512/767/1280 interfaces are selectable as devices 2/3/4 respectively, from the device selection menu. The 1280 interface (device 4) is available only in the VAX implementation of SMP.

6.1.3.4 INTELLIGENT SYSTEMS CORPORATION (ISC) 8001R

The ISC 8001R is a color raster terminal used in a serial, RS-232C environment. The ISC device level subroutine library was developed specifically for LaRC by Computer Sciences Corporation.

The screen/raster space limits shown for the ISC in TABLE 7 indicate that it is a low-resolution display device. Interactive graphics input and output of graphics text are not supported. The super pixel capability of the ISC is used by the device interface to output a hidden surface rendering. It allows for the display of 4x4 blocks of pixels in any of 4000 distinct colors.

The ISC interface is selectable as device 4 from the device selection menu. This interface is available only in the PRIME implementation of SMP.

6.1.3.5 TEKTRONIX 4100 SERIES

The TEKTRONIX 4100 series terminals are used with SMP in a serial, RS-232C environment. The TEKTRONIX device-level subroutine library which supports these devices was developed specifically for LaRC by Computer Sciences Corporation.

Each of these devices has an additional area of refresh memory called the dialog scroll area which is used by SMP for messages and prompts. The dialog area scroll is enabled and the dialog area is not erased when the graphics area is erased. The user may control the visibility of the dialog area locally.

The 4107/4109 terminals are medium resolution (see TABLE 7) color raster devices which support 16 displayable colors. The hidden surface dithering option should be applied when attempting a shaded rendering to achieve satisfactory results. Interactive graphics input is obtained via the joystick.

The 4115 model is an interactive terminal that uses a high resolution (see TABLE 7) color monitor. The number of bit planes available on the device is a function of the option purchased and may be either 4, 6, or 8. This controls the number of displayable colors available. If less than 8 bit

planes are available, the dithering option should be applied to obtain an acceptable hidden surface rendering. The device-level interface queries the device for the number of bit planes. Interactive graphics input is obtained via the thumbwheel cursor.

While the 4129 model has more capability than the 4115 and may be used in DMA mode, the device-level interface utilized by SMP is the same for the 4129 as it is for the 4115.

6.1.3.6 TEKTRONIX 4010 SERIES

The TEKTRONIX 4010 model terminals are monochrome storage-tube CRTs used in a serial, RS-232C environment. The addressable graphics area is 1024 by 780 points as denoted in TABLE 7. The device interface is supported in the modeller with the TEKTRONIX PLOT-10 TERMINAL CONTROL SYSTEM (TCS) subroutine library.

Interactive graphics input is supported via the thumbwheel cursor. Hidden surface renderings should not be attempted since raster operations are not supported by this type of device.

6.2 USER ENVIRONMENT

SMP's user environment is described in the following sections. Since it varies slightly from PRIMOS to VMS, each host is discussed separately.

6.2.1 VAX/VMS ENVIRONMENT

Once "logged" into VMS, SMP is executed from the command procedure EXEC.COM as follows:

```
@ [CSC.SMP.VER4]EXEC
```

For commands that require a full file name as input, READ (see section 4.1) or WRITE (see section 4.2), a VMS "pathname" is acceptable input.

The modeller first prompts the designer for the terminal type:

```
ENTER DEVICE CODE
  0 = NON-GRAPHICS TERMINAL
  1 = TEKTRONIX 401X SERIES
  2 = AED 512
  3 = AED 767
  4 = AED 1280
  5 = TEKTRONIX 4100 SERIES
  6 = ADAGE 3000
  7 = ADAGE 3000 (SOLID)
```

The prompt is repeated until a legal response is encountered.

The next prompt

```
ENTER BAUD RATE
  1 = 9600
  2 = 1200
```

is included to accommodate "remote logins". Again, this prompt is repeated until a legitimate response is received.

The SMP output is a title page, shown here for the alphanumeric terminal display device.

SOLID MODELING PROGRAM
DEVELOPED BY :

COMPUTER SCIENCES CORP.
HAMPTON, VIRGINIA
DEVELOPED FOR :

NASA LARC

<CR> TO CONTINUE

The user enters <CR> to continue. At this juncture the designer is positioned at the SMP command level and is prompted accordingly with

ENTER COMMAND>

The major command classes are detailed in section 4. Recall that information regarding the command levels, command sublevels, subcommands and command options can be obtained through the on-line HELP facility triggered by the character "H". "Help" is available from the command level and the major command sublevels. An illegal response from a command level or command sublevel results in a menu of legal commands being displayed on the user's terminal.

In a typical session the user will enter either the READ (see section 4.1) or EDIT (see section 4.3) command sublevels. If

a model has been created earlier in SMP or from some external source, the READ command should be selected. EDIT should be selected if a model is to be built from "scratch".

The majority of the session will involve transferring between the EDIT, DISPLAY and ANALYSIS (see section 4.4) commands to properly construct (i.e. "edit") and orient the model geometry and compute mass properties.

When the editing process is complete, the WRITE command (see section 4.2) is normally selected to save the edited model geometry.

Once the edited model is saved, the user can continue the editing process or "exit" the system with the QUIT command.

The above discussion briefly sketches a typical interactive session with the modeller. An experienced user can naturally tailor the command selection to meet his/her requirements.

6.2.2 PRIME/PRIMOS ENVIRONMENT

Once "logged" into PRIMOS [14], SMP is executed with the PRIMOS SEQ utility as follows:

```
SEQ CSC>SMP
```

For commands that require a file name as input, READ (see section 4.1) or WRITE (see section 4.2), a PRIMOS "pathname" is acceptable input.

The modeller first prompts the designer for the terminal type:

```
ENTER DEVICE CODE
0 = NON-GRAPHICS TERMINAL
1 = TEKTRONIX 401X SERIES
2 = AED 512
3 = AED 767
4 = ISC 8001R
5 = TEKTRONIX 4100 SERIES
```

From this point on SMP functions as described in section 6.2.1. Exceptions are noted in section 6.1.2.1.

APPENDIX A. SMP IMPLEMENTATION

The purpose of this appendix is to provide an implementor with information concerning the structure and organization of the SMP software. Although the target computer system for this discussion is a VAX 11/785 running VMS 4.2, much of the information presented is "host" independent.

SMP is comprised of 13 software modules supported by a collection of utility and graphics device libraries. The major software components are listed in alphabetical order and briefly described in TABLE 8. The utility and device libraries are similarly presented in TABLE 9.

TABLE 8

SMP SOFTWARE MODULES

<u>NAME</u>	<u>DESCRIPTION</u>
ALFAED	The alphanumeric part editing (EDIT) software module
DIDGGS	The graphics device interface modules. New devices are interfaced by modifying this code.
DISPLAY	Module to control and perform all graphics DISPLAY functions.
GRAFED	The graphical part editing software module.
GVS	General Vehicle Synthesizer module.
MANALYS	Module to compute the mass properties and projected areas.
MISCCMD	Miscellaneous commands such as part "dimensioning" reside in this module.
PF2GF	Manipulates the data structure necessary to insure compatibility between the parts-file and the geometry-file.
PRIMTV	Module for generating geometry for the basic primitive parts. There is a separate module for each primitive.
PRIORTY	Hidden surface processing algorithm resides in this module.
SCANLINE	Module which renders both hidden surface and hidden line images.
SETOP	Code to perform the boolean operations on two designated parts.
SMPMISC	Main driver with support utilities for all commands. The READ and WRITE commands reside in this module.

TABLE 9

SMP UTILITY AND DEVICE LIBRARIES

<u>NAME</u>	<u>DESCRIPTION</u>
ADAGE_DUM	"Dummy" library for the Adage 3000 used to satisfy externals during linking process.
AED	AED device driver library.
CSCLIB	Low-level device I/O utility library.
ISC	ISC device driver library.
LASSLIB	RIM interface utility library required only with GVS.
PLOT/10	Tektronix PLOT-10 Terminal Control System (TCS) library.
RIMLIB	RIM utility library including FORTRAN interface. Required only when exercising GVS.
TEK4100	Tektronix 4100 device driver library.
ULIB	Utility library for performing 3-D transformations with matrices.

Source code is provided for all libraries except RIMLIB and PLOT-10. The implementing installation is responsible for acquiring these two packages. Only "dummy" routines are provided for the Adage 3000 interface. If GVS is not required, LASSLIB and RIMLIB are unnecessary.

The individual software modules may be compiled under VMS by invoking FOR without special options. FIGURE 18 provides a sample DCL command file for linking all the required modules and libraries. Consult sections 6.1.1 and 6.1.2 before installing SMP on a "host" other than a VAX running VMS.

LINK /DEBUG/EXECUTABLE = SMP.EXE -

\$! SMP PROGRAM MODULES

\$! SMPMISC -

+ PF2GF -

+ PRIMTV -

+ SETOP -

+ ALFAED -

+ DISPLAY -

+ PRIORTY -

+ SCANLINE -

+ GRAFED -

+ MISCCMD -

+ MANALYS -

+ GVS -

+ DIDDGS -

\$! (3-D) TRANSFORMATION UTILITY LIBRARY

+ [CSC.LIB]JULIB/LIBRARY -

\$! GRAPHICS DEVICE DEPENDENT MODULES

+ [CSC.LIB]JISC/LIBRARY

+ [CSC.LIB]JAED/LIBRARY

+ [CSC.LIB]TEK4100/LIBRARY -

\$! HOST DEPENDENT SUBROUTINE LIBRARY

+ [CSC.LIB]CSCLIB/LIBRARY -

\$! ADAGE DUMMY LIBRARY FOR SATISFYING EXTERNALS

+ [CSC.SMP.VER4]ADAGE_DUM/LIB -

```
$!      IDEAS UTILITY LIBRARY FOR RIM ACCESS
+ [CSC.LIB]LASSLIB/LIB -
$!      PLOT10 TCS LIBRARY
+ [LIBS]PLOT10/LIBRARY -
$!      RIM DATA BASE FORTRAN INTERFACE LIBRARY
+ [LIBS]RIMLIB/LIBRARY
```

FIGURE 18

LINKING SMP

APPENDIX B. TRUSS FORMATION

This discussion details how the boundary representation of a truss is formed. In the construction of the boundary representation of a truss, nodes and faces are conserved. That is, duplicate nodes and faces have been eliminated. An additional savings may be realized by defining the "truss" in such a manner that the maximum number of bays required in any direction is in the vertical direction. The following example illustrates this point: When $NVB=1$, $NHB=1$, and $NDB=10$, the number of points and faces required to define the boundary representation are 344 and 380, respectively. By specifying $NVB=10$, $NHB=1$, and $NDB=1$, the number of points is reduced to 272, a savings of 21 percent, and the number of faces is reduced to 200, a savings of nearly 48 percent.

Creation of the boundary representation for the vertical supports is undertaken first and is accomplished through determination of eight vertices such that the length along the x-axis is the length of a truss element (L), and the width along the z-axis is the width of a truss element (W). The height along the y-axis is computed as a function of the number of vertical bays (NVB), the vertical distance between supports (DV), and the height of a truss element along the y-axis (H) as follows:

$$\text{HEIGHT} = H * (\text{NVB} + 1) + \text{DV} * \text{NVB}$$

Six planar subsections are formed for each vertical support. The first vertical support formed is positioned such that its lower left corner is at (0,0,0). As noted earlier, this is the left corner for the "truss". Subsequent vertical supports are positioned first by incrementing the x-coordinate of the lower left corner by (L + DH) until (NHB + 1) vertical supports are formed and then reinitializing the x-coordinate and decrementing the z-coordinate of the lower left corner by (W + DW) until (NDB + 1) * (NHB + 1) vertical supports have been formed. The boundary representation for each horizontal support is undertaken next and is accomplished through determination of eight vertices such that the length along the x-axis is the horizontal distance between truss elements (DH), the height along the y-axis is the height of a truss element (H), and the width along the z-axis is the width of a truss element (W). Vertices which coincide with vertical support vertices are obtained from the boundary representation for the vertical supports. This generally occurs for the "bottom" and "top" horizontal supports.

Four planar subsections are formed for each horizontal support. The two faces on each support which abut vertical

supports are not formed. The first horizontal support is positioned such that its lower left corner is at $(L, 0, 0)$. Subsequent horizontal supports are positioned first by incrementing the y-coordinate of the lower left corner by $(H + DV)$ until $(NVB + 1)$ horizontal supports are formed. The y-coordinate of the lower left corner is then reinitialized and the x-coordinate is incremented $(L + DH)$. Another $(NVB + 1)$ horizontal supports are formed, and the process continues until $(NVB + 1) * NHB$ horizontal supports have been formed. The x- and y-coordinates of the lower left corner are then reinitialized and the z-coordinate is decremented by $(W + DW)$. Another $(NVB + 1) * NHB$ horizontal supports are formed and the process continues until $(NVB + 1) * NHB * (NDB + 1)$ horizontal supports have been formed.

The boundary representation for each depth support is computed last and is again accomplished through determination of eight vertices. The length along the x-axis is the length of a truss element (L) , the height along the y-axis is the height of a truss element (H) , and the width along the z-axis is the depth distance between truss elements (DW) . Vertices which coincide with vertices already computed for vertical or horizontal supports are not recomputed but rather are obtained from the boundary representation of those supports. This generally occurs everywhere except along the "left" and

"right" perimeter of the "truss". In these cases, at least two and at most four vertices are computed.

Four planar subsections are formed for each depth support. The two faces which abut vertical supports are not formed. The first depth support formed is positioned such that its lower left corner is at $(0,0,W)$. The y-coordinate of the lower left corner of each depth support is then incremented by $(H + DV)$ until $(NVB + 1)$ depth supports are formed. The y-coordinate is then reinitialized and the x-coordinate is incremented by $(L + DH)$. Another $(NVB + 1)$ depth supports are formed by incrementing the y-coordinate NVB times. This is done until $(NVB + 1) * (NHB + 1)$ depth supports are formed. The x- and y-coordinates are then reinitialized and the z-coordinate is decremented by $(W + DW)$. This process continues until $(NVB + 1) * (NHB + 1) * NDB$ depth supports have been formed.

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16. Abstract This paper documents a prototype solid modeling program, SMP, developed by CSC for Langley Research Center (LaRC). Version 2 of the SMP software is employed by the Spacecraft Analysis Branch (SAB) of the Space Systems Division (SSD) for preliminary space station design, but is intended as a general purpose tool. The SMP document provides details concerning: the basic geometric modeling primitives and associated operators, the data representation scheme utilized to structure the geometric model, the available commands for both editing, displaying, and mass properties analysis of the solid model, the interactive user interface and the input/output interfaces to external software, and the utility of the package in the LaRC computing environment. The document is sufficiently detailed to serve both as a user's guide and reference manual.					
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